

Analysis and Management of Temperature Fields in F1 Cars



Christopher Lim
University College
University of Oxford

A thesis submitted for the degree of

Doctor of Philosophy

Michaelmas Term 2017

Dedicated to Alan Edward Oram.

Without whom I would never have discovered the joy of engineering.

Abstract

Analysis and Management of Temperature Fields in F1 Cars

This thesis investigates the broad subject of thermal management problems currently encountered in Formula One race car design.

A computationally economical tool, based on linear superposition, for predicting the temperature field arising from a set of thermal and inlet velocity boundary conditions was developed. Using a set of base analyses, the research showed that it is possible to superpose and scale these results in order to predict the temperature field for differing sets of boundary conditions. This method was shown to have a significant speed advantage over typical computational simulations. An experimental facility was designed and built to provide validation for aspects of the linear superposition approach.

A method of measuring the cylinder wall heat flux has been developed using thin film gauge technology. The resulting sensor was designed to fit the mounting of existing instrumentation in order to avoid requiring large scale modifications to existing test facilities. The design makes use of modern rapid prototyping techniques in order to meet this mounting requirement and to provide a novel solution to routing the signal from the thin film gauge.

In addition, the research investigated a method for predicting the cylinder wall temperature in real-time. The cylinder wall is subject to heat fluxes from in-cylinder gases during the engine cycle on the inner face and the effect of the coolant jacket on the outer face. Two separate methods were used to process these thermal boundary conditions respectively, before being superposed in order to form the whole solution. The computation time of the method is characterised in order to demonstrate its feasibility for real-time operation.

Acknowledgements

I would like to thank everyone that has offered help and support to me throughout my time at the Osney Lab.

Firstly, I'd like to thank Professor Peter Ireland for supervising me through the past 4 years. Your knowledge and enthusiasm for the subject have been integral to the completion of this work and the fact that I loved every minute of my time in the Osney Lab.

This body of work was funded by Ferrari S.p.A. and I'm incredibly grateful for their support throughout the project. I'd also like to thank Nick Collett and his colleagues, Ioannis Kitsopanidis and Rhodri Moseley, for giving me the opportunity to work with such an amazing company and team of engineers. In particular, I'm indebted to Nick for his boundless enthusiasm and effort in supporting this project from the very beginning.

Thank you to all those staff and students at the Osney Lab in getting my experiments off the ground. Particular gratitude to Duncan Blake, Mike Japp, and Greg King for putting up with my constant requests and best attempts at breaking your creations.

In addition, thank you to all my friends and family that have supported me throughout my DPhil. I've immensely enjoyed my time in Oxford and that is because of you. Special thanks also to Emily for providing encouragement, support, and levity throughout the past 4 years. It has been appreciated more than you know.

Lastly, I'd like to thank my parents who have given me everything I needed and more. Without your support, I would never have been where I am today. Thank you.

Contents

1	Introduction	1
1.1	Origins of Thermal Loads in Formula One	1
1.1.1	Temperature effects on race car reliability and robustness . . .	3
1.1.2	Component temperature effects on race car performance . . .	4
1.2	Current Thermal Design Challenges in Formula One	5
1.2.1	Requirement for short design timescales	5
1.2.2	Restrictions on permitted computational simulations	6
1.3	Use of Experiments in Formula One	8
1.3.1	Development of race car aerodynamics	8
1.3.2	Use of experiments in power unit development	9
1.4	Computational Analysis in Formula One	10
1.5	Aims of Current Thesis	11
1.6	Research Contributions	12
1.7	Overview of Thesis Structure	14
1.8	Publications	15
2	Literature Review	17
2.1	Introduction to Literature Review	17
2.2	Thermal Analyses of the Underhood Region	18
2.2.1	The importance of characterising underhood thermal behaviour	18
2.2.2	Current design methodology for the underhood environment .	19
2.3	Use of Linear Superposition for Thermal Analyses	22
2.4	Investigations of In-Cylinder Thermal Conditions	24
2.4.1	Introduction	24
2.4.2	Empirical models to predict in-cylinder heat transfer	25

2.4.3	Validation of empirical heat transfer models	29
2.4.3.1	Experimental validation	30
2.4.3.2	Computational validation	33
2.4.4	Calculating heat release from pressure measurements	34
2.5	Spark Advance Optimisation	36
2.5.1	Introduction	36
2.5.2	Spark advance control	37
2.5.3	Calculation of spark advance values	39
2.5.4	Retardation of spark advance and engine knock	39
2.6	Conclusions	40
3	Linear Superposition Theory	42
3.1	Introduction	42
3.2	Linear Superposition of the Convection Transfer Equation	42
3.3	Heat Transfer Dimensional Analysis	43
3.4	Implementation of the Linear Superposition Method	45
3.4.1	Definition of the temperature influence matrix	45
3.4.2	Determination of the temperature influence matrix	46
3.4.3	Discretisation of continuous domains	47
3.5	An Approach to Account for Varying Flow Velocities	48
3.6	Application of a NTU- ϵ Correction Factor	50
3.6.1	Heat exchanger analysis	50
3.6.2	Stanton number as a function of Reynolds number	52
3.7	Linear Superposition using Constant Temperature Boundary Conditions	53
3.8	Temperature Dependent Non-Linear Effects	54
3.8.1	Changes in working fluid properties	54
3.8.2	Natural convection effects	54
4	Linear Superposition Computational Results	56
4.1	Computational Model	56
4.1.1	Computational mesh	56
4.1.2	Solver conditions and post processing	57
4.2	Linear Scaling of Temperature Fields	59

4.2.1	Populating the temperature influence matrix	59
4.2.2	Superposition of individual temperature fields	61
4.2.3	Scaling of heat fluxes	64
4.3	The Effect of Changing Inlet Velocity	68
4.3.1	Linear mass flow scaling	69
4.3.2	Scaling using Stanton number dependency	73
4.3.3	Temperature gradient weighted Stanton number scaling	77
4.4	Investigation of Velocity Scaling on Alternative Geometry	84
4.5	Computation Time	86
5	Experimental Studies	88
5.1	Introduction	88
5.2	Experimental Test Section	89
5.3	Internal Assembly	90
5.4	Heat Flux Addition	91
5.4.1	Electrical heater pads	91
5.4.2	Electrical heater mesh	92
5.5	Instrumentation	93
5.5.1	Point temperature and pressure measurements	93
5.5.2	Measurements using thermochromic liquid crystals	94
5.5.2.1	Thermochromic liquid crystals	96
5.5.2.2	Thermochromic liquid crystal calibration	97
5.5.2.3	Imaging equipment	97
5.5.2.4	Image mosaicing	98
5.5.2.5	Cross sectional temperature measurements	99
5.5.2.6	Heat transfer coefficient calculation	99
5.6	Experimental Results	101
5.6.1	Steady state temperature measurements	101
5.6.1.1	Temperature rake measurements	101
5.6.1.2	Rake 1 results and discussion	102
5.6.1.3	Rake 2 results and discussion	103
5.6.1.4	Cross-sectional temperature results	108
5.6.2	Transient heat transfer experiments	110

5.6.2.1	Thermochromic liquid crystal experimental method .	110
5.6.2.2	Validation of thermochromic liquid crystal results . .	110
5.6.2.3	Computational model validation using heat transfer coefficients	111
5.7	Summary of Linear Superposition Model Development	115

6 Application of Thin Film Gauge Technology to In-Cylinder Heat

	Flux Measurements	116
6.1	Introduction	116
6.2	Traditional Heat Flux Measurement Techniques	116
6.3	Thin Film Gauge Technology	117
6.3.1	Calculation of heat flux from surface temperature	119
6.3.1.1	Single layer heat transfer gauge	119
6.3.1.2	Two-layer heat transfer gauge	120
6.3.2	Impulse response processing in transient heat transfer experi- ments	122
6.4	Calibration of Thin Film Gauges	124
6.5	Base Design for Heat Flux Sensor	125
6.5.1	Base geometry	125
6.5.2	Cable routing	126
6.6	Design and Manufacture of a Two-Layer Heat Flux Sensor	127
6.6.1	Decision factors for coating choice	127
6.6.2	Insulator coating materials	128
6.6.3	Initial observations on platinum and gold deployment	129
6.6.4	Calibration of gauges on test disc	131
6.6.5	Coating of the probe geometry	134
6.6.5.1	Application of vitreous enamel to probe geometry . .	134
6.6.5.2	Application of alumina-zirconia coating to the base geometry	137
6.7	Design and Manufacture of a Single Layer Heat Flux Sensor	139
6.7.1	Determination of base material	139
6.7.2	Initial observations on platinum and gold deployment	140
6.7.3	Optical and scanning electron microscopy of samples	141

6.7.4	Calibration of thin film gauges on ceramic substrate	143
6.7.5	Use of ceramics in heat flux sensor geometry	143
6.8	Effect of Annealing on Platinum Thin Film Gauges	145
7	Thin Film Gauge Experimental Data	147
7.1	Response Time Testing of Example Thin Film Gauges	147
7.2	Engine Testing	149
7.2.1	Comparison with empirical heat transfer models	150
7.2.2	Cold start conditions	153
7.2.3	Firing condition	154
7.3	Thin Film Gauge Robustness	156
7.4	Application of an Amorphous Diamond Coating	158
7.4.1	Introduction to diamond shielded thin film gauges	158
7.4.2	Amorphous diamond application method	158
7.4.3	Calculation of heat fluxes from a shielded thin film gauge . . .	161
7.4.4	Results of applying an amorphous diamond coating	161
7.5	Summary of experimental findings	163
8	Optimising Spark Advance using Wall Temperature	166
8.1	Introduction	166
8.2	Proposed System for Predicting Cylinder Wall Temperature	166
8.3	Superposition of Combustion and Coolant Cases	168
8.3.1	Boundary conditions for superposition	170
8.4	Use of the FFT Method to Predict Cylinder Wall Temperature	171
8.4.1	Introduction to the FFT method	171
8.4.2	Calculation of the impulse response function using FEA simulations	172
8.5	Prediction of Coolant Effects using Finite Differencing	174
8.5.1	Effect of the coolant on cylinder wall temperature	174
8.5.2	Calculation of the temperature profile through the cylinder wall	174
8.5.3	Boundary conditions for finite difference model	176
8.6	Time Marching Algorithm for Real-Time Implementation	177
8.6.1	Introduction	177

8.6.2	Implementation of the FFT method in real-time	178
8.6.3	Structure of FFT real-time implementation	181
9	In-Cylinder Surface Temperature Computational Results	184
9.1	1-D Conduction Model	184
9.1.1	Computational model description	184
9.1.2	Mesh independence study	186
9.1.3	FFT calibration error sensitivity	187
9.1.4	Calculation of heat flux results	188
9.1.5	Calculation of coolant effects	191
9.1.5.1	Model tuning using gradient descent	192
9.1.6	Superposition of Results	193
9.2	Cylinder Cross Section Results	195
9.2.1	Computational model description	195
9.2.2	Calculation of heat flux results	196
9.2.3	Calculation of coolant effects	198
9.2.4	Superposition of results	200
9.2.5	Continuous implementation of cylinder wall model	202
9.3	Computation Time	205
9.3.1	Setup and tuning time	205
9.3.2	Processing time	206
9.3.3	Comparison to 3-D FEA simulations	207
9.4	Summary of In-Cylinder Surface Temperature Model Development . .	207
10	Summary and Conclusions	209
10.1	Predicting temperature fields using linear superposition	210
10.1.1	Computational Work	210
10.1.2	Experimental Work	211
10.2	Measurement of heat fluxes inside an internal combustion engine . . .	213
10.3	Predicting cylinder wall temperature using fast Fourier transform method and finite difference method	214
10.3.1	Use of fast Fourier transform method to predict temperature .	215
10.3.2	Use of finite differencing to calculate coolant effect	216

10.4	Recommendations for future work	216
10.4.1	Linear superposition method	216
10.4.2	Internal combustion engine heat flux measurement	218
10.4.3	Cylinder wall temperature prediction using a tuned thermal model	219
Bibliography		220
A Mass Flow Scaling Results for Convergent Geometry		232
B Thin Film Gauge Calibration Results		235
B.1	Calibration of vitreous enamel coated test disc	235
B.2	Calibration of alumina-zirconia coated test disc	235
B.3	Calibration of ceramic substrate test samples	236
B.4	Calibration of engine probes	236
C Uncertainty Analysis		237
C.1	Dimensionless Temperature Coefficient	238
C.2	Heat Transfer Coefficient Measurements	239
C.3	Surface Temperature Measurements using Thin Film Gauges	240

Nomenclature

Where variables overlap, the relevant chapter is listed in brackets. Variables may have a specific use within a chapter that differs from the list below and will be defined in context.

Roman

a, b, c	Empirical coefficients
A	Surface area (m^2)
B	Cylinder bore (m)
c_p	Specific heat capacity at constant pressure ($\text{J kg}^{-1} \text{K}^{-1}$)
D	Characteristic length (m)
g	Gravitational acceleration (m s^{-2})
h	Heat transfer coefficient ($\text{W m}^{-2} \text{K}^{-1}$)
I	Electrical current (A)
k	Thermal conductivity ($\text{W m}^{-1} \text{K}^{-1}$)
L/D	Length to diameter ratio
$\dot{m}$	Mass flow rate (kg s^{-1})
p	Pressure (Pa)
q	Heat flux (W m^{-2})
Q	Heat flow (W)
R	Electrical resistance (Ω)
t	Time (s)
T	Temperature (K)
u	Gas velocity (m s^{-1})
U	Internal energy (J)
v_p	Piston velocity (m s^{-1})

$v_{p,m}$	Mean piston velocity (m s^{-1})
V	Cylinder volume (2, 6, 7) (m^3)
V	Voltage (Appendix C) (V)
V_s	Swept volume (m^3)
w	Weighting for temperature influence coefficients
W	Work (J)

Subscripts

∞	Mainstream conditions
0	Initial condition
0	Reference condition (6, Appendix C)
c	Coolant
g	Gas
i	At a point i in the domain
in	Inlet condition
m	Motored condition
$q - T$	Heat flux to temperature
r	Reference condition
s	Surface
$T - q$	Temperature to heat flux
w	Cylinder wall

Greek

α	Thermal diffusivity ($\text{m}^2 \text{s}^{-1}$)
α_{tc}	Temperature coefficient of resistivity (K^{-1})
β	Coefficient of thermal expansion (K^{-1})
ϵ	Uncertainty
η_c	Convective efficiency
γ	Ratio of specific heats
λ	Finite-difference form of Fourier number
ω_c	Crankshaft rotational speed (rad s^{-1})
μ	Dynamic viscosity (Pa s)

Φ	Dimensionless temperature parameter
ψ	Temperature influence coefficient (KW^{-1})
ρ	Density (kg m^{-3})
θ	Crank angle (deg)
ξ	Dimensionless temperature influence coefficient
ζ	Temperature influence coefficient (for temperature boundary conditions)

Non-Dimensional Numbers

Fo	Fourier number
Gr	Grashof number
Nu	Nusselt number
Pr	Prandtl number
Re	Reynolds number
St	Stanton number

Chapter 1

Introduction

1.1 Origins of Thermal Loads in Formula One

A Formula One race car is subject to a large number of loads which vary around the track. A complete understanding and management of this loading is key to the successful design of the car. The subject of this thesis is an investigation into the thermal loads originating from the power unit of the car. As the power unit is often operating at the peak of its capability in order to maximise the performance of the race car, accurate analysis and successful management of these thermal loads throughout are key to ensuring both the robustness of the race car and its ability to operate at maximum performance for the duration of a race.

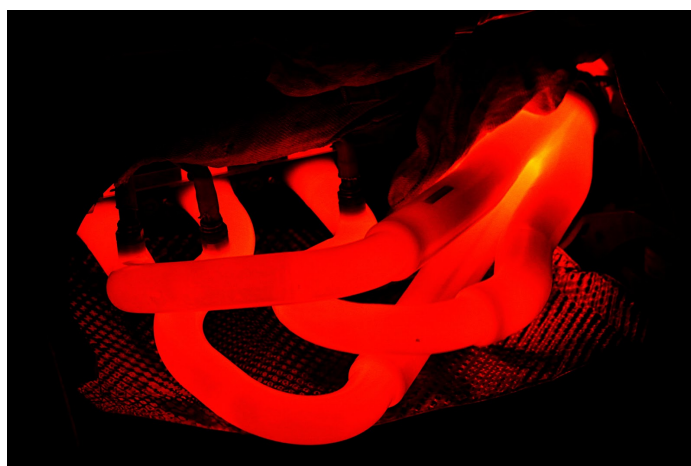


Figure 1.1: Renault F1 V6 engine on test bench [1]

At the time of writing, Formula One cars are powered by a hybrid power unit, comprising a 1.6 litre V6 turbocharged internal combustion engine combined with electric motor generator units coupled to the turbocharger shaft (known as the motor generator unit heat, or MGU-H) and the main drive shaft (known as the motor generator unit kinetic, or MGU-K). Together, the power unit will output up to 650kW of power to the drive train (though the peak power is not accessible throughout the whole race).

The majority of this power comes from the internal combustion engine, which will provide approximately 500kW to the drive train at peak output. Whilst the thermal efficiency of the internal combustion engine in Formula One is far better than that of a consumer vehicle, a Formula One race car will often output its power to the drive train at a peak of approximately 45% thermal efficiency, which results in significant amounts of heat that must be effectively dissipated.

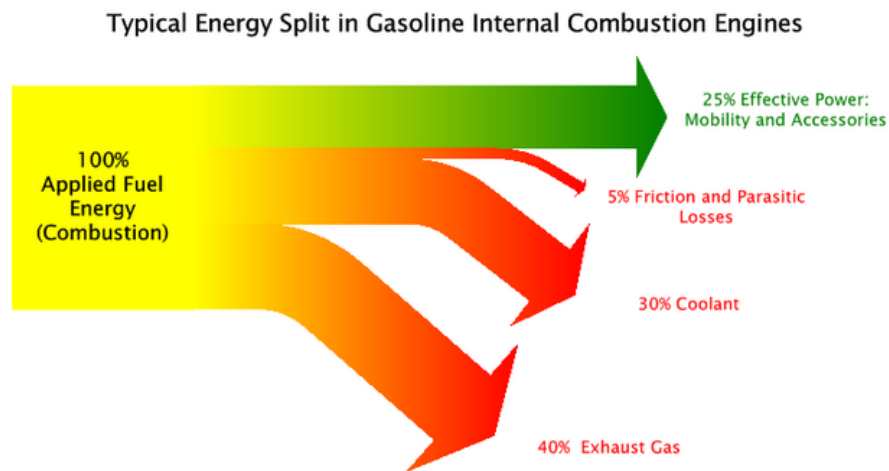


Figure 1.2: Sankey diagram of energy conversion in a typical consumer internal combustion engine [2]

This effectively results in a large quantity of power, released from the combustion process, that is dissipated throughout the cooling system or released as enthalpy in the exhaust gases. A large proportion of the former will be thermal energy that is dissipated in the underhood region (see Figure 1.2 for the related example of a consumer vehicle).

This excess released energy must be effectively and efficiently dissipated in order to maintain the integrity of the underhood components. The air flow field around the power unit in the underhood region is complex, owing to the intricate geometry that is constrained by both aerodynamic considerations and governing body regulations. The combined geometrical and resulting flow field complexity make investigation and management of the thermal loads in this region a research challenge. Thorough understanding of the thermal loads on the race car, throughout the different operating points that occur during a race weekend, is essential to enable engineers to reach an optimal design.

1.1.1 Temperature effects on race car reliability and robustness

The performance of the majority of components in the underhood region is temperature dependent and the lifetime of each component is also usually temperature sensitive. As part of a recent package of cost-cutting measures, the FIA regulations have dictated that each race car only has access to a certain number of power unit components per season (with racing penalties for any additional components used). This means, since each power unit component must last a certain number of races, that robustness of the power unit components can also be a major factor in determining success throughout the season.

In addition to this, the combustion unit is often operating at the peak of its thermal limits (the point at which combustion is knock-limited) in order to output the maximum amount of power to the drive train. As such, the peak gas temperatures in the combustion chamber often exceeds the capabilities of the materials used. Thus, the cooling strategy and thermal management of the power unit components are vital to ensure reliability throughout each race.

This is compounded by the material properties of most power unit components being temperature dependent. As non-uniform temperature distributions engender thermal stresses, understanding the heat flows within the system are essential to determining the lifespan of each component. Given that the power unit is operating on the very limit of performance, engine failures are more common in Formula One than the av-

erage consumer vehicle and have often been crucial in ultimately deciding the winner of championships.

The key to establishing robustness is an accurate and in-depth understanding of the thermal loads that occur throughout a race weekend and how they vary through the operating range of the race car. Once this understanding has been established, an effective thermal management solution can then be devised in order to best manage these thermal loads within the restrictions imposed by both the FIA regulations and aerodynamic considerations.

1.1.2 Component temperature effects on race car performance

The performance of many components on a Formula One car are heavily temperature dependent and as such, one key aspect in the design of Formula One race cars is management of the various heat loads such that each component is operating in an environment as close to its optimum temperature as possible.

The power unit is no exception to this with both the power output and efficiency being heavily temperature dependent. Much of the engine specification (such as number of cylinders, cylinder bore, and compression ratio) is governed by the FIA Governing Body regulations [3]. Along with these restrictions, the total fuel carried (with no refuelling allowed) and fuel flow rate into the combustion engine are also both limited by the regulations which places a calorific limit on the input to the combustion unit which is standardised across the teams. As a result of this, the efficiency of the power unit has become increasingly important for two reasons.

Firstly, with no refuelling allowed, the power unit must be efficient enough to finish the race within the total fuel limit set out by the regulations. In addition to this, with the calorific input to the combustion unit becoming standardised amongst the teams, the efficiency of the respective power unit used by each team dictates how much power is available at the drive train. Hence, teams using more efficient power units will gain a significant advantage in terms of available deliverable power. This has led to the efficiency of the respective power units being the key performance differentiator between the competition in recent seasons.

Secondly, in order to provide maximum power to the drive train, the power unit operates at a knock-limited condition. As the cooling strategy and management of the heat produced is key to managing the operating environment of the power unit components, an efficient cooling strategy allows this limit to be raised. By raising the limit at which engine-knock occurs, the power unit is able to run at increased temperatures during the combustion process which allows more work to be extracted from the engine cycle.

Whilst it is important to have a cooling strategy and management that will adequately manage the thermal loading on the power unit, it is equally important to consider the effect of over-designing the cooling strategy on the performance of the race car. Formula One design can be considered as a constant optimisation problem whereby the goal is to maximise the performance of the race car with respect to the components being able to survive a set number of race weekends. One consideration in this optimisation scenario is that the over-design of the cooling strategy will be detrimental to the overall car performance due to the correlation between increased coolant air mass flow rate, air flow total pressure reduction, and aerodynamic drag from the radiator.

1.2 Current Thermal Design Challenges in Formula One

The investigation and management of thermal loads within a Formula One race presents a large number of technical and logistical challenges for design engineers. As previously mentioned, the successful management of the various thermal loads are often crucial to the race car operating at its maximum capability and its chances of success in any given race. The aforementioned challenges revolve around two core themes.

1.2.1 Requirement for short design timescales

The lifetime of any given race car design within Formula One is relatively short when compared to consumer vehicles. Large amounts of the race car are often changed in

order to meet the constantly evolving and detailed regulations from the FIA Governing Body who often revise the rules and regulations on an annual basis in order to factor in the lessons learned during each season of racing.

While the core design of the race car may remain stable throughout the season, parts of the car will still constantly change and evolve on a race by race basis in order to incorporate gradual aerodynamic updates which aim to either improve the base aerodynamic performance of the race car or to tailor the aerodynamics of the car to best fit the downforce characteristics of a certain race track. This constant design evolution places unique time pressures on the design process of the race car as upgrades often need to be designed, tested, manufactured, and fitted on a weekly or fortnightly basis.

In addition to this time pressure, the aerodynamic performance of the car is often prioritised over the thermal design of the car as this often has a larger effect on the overall performance of the race car. As mentioned previously, because the design of the race car will constantly evolve throughout the season, the coolant strategy must also be flexible in terms of both geometry and performance so as not to impose any restrictions on any future aerodynamic or component evolution of the race car.

In addition to the aerodynamic restrictions on the geometry, there are restrictions on both geometry and weight imposed by the FIA Governing Body regulations [3]. With the thermal loading often set by the specifications and performance of the power train, the base requirement for any given thermal management strategy is that it must allow the power train to perform reliably within the constraints imposed by the aerodynamic design and the governing body regulations.

1.2.2 Restrictions on permitted computational simulations

As part of a package of cost-cutting initiatives (designed to reduce the barriers to entry for new teams) introduced in 2013, the current FIA Governing Body regulations [4] place a restriction on the amount of computational power and wind tunnel time that each team can use in a given season. The system works on a sliding scale, whereby a team that uses more computational power has less wind tunnel time available and vice versa (see Figure 1.3).

FIA Governing Body Restrictions on CFD Simulations and Wind Tunnel Testing

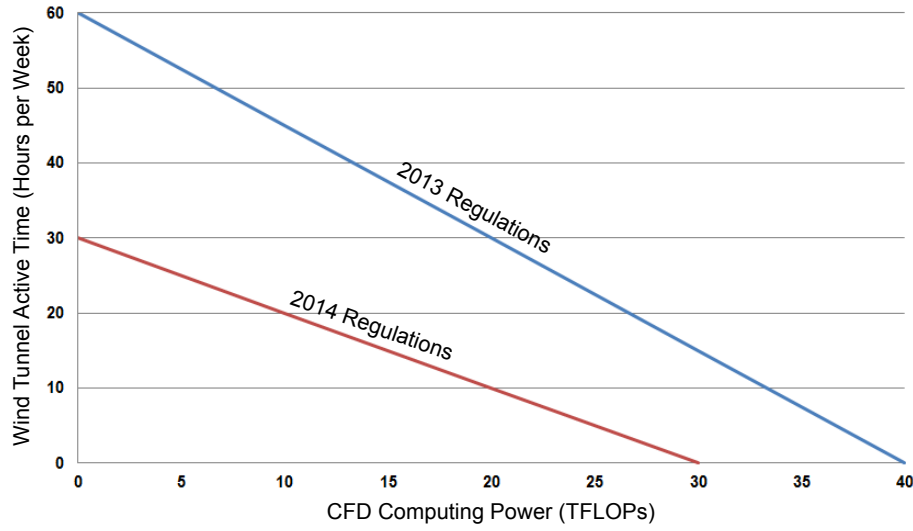


Figure 1.3: FIA Governing Body restrictions on CFD simulations and wind tunnel testing. Teams are required to stay to the left of the lines shown.

With these restrictions on both computational simulations and wind tunnel time, teams are required to prioritise the development of certain aspects of the car over others. As thermal analyses require the same computational resources as aerodynamic investigations, they are subject to the same restrictions. As alluded to earlier, aerodynamic development is often considered to be more important than thermal investigations due to the greater impact of aerodynamic developments on overall car performance. While it was noted that over-design of the cooling strategy can be detrimental to overall car performance, this is often disregarded compared to the advantages that can be gained through aerodynamic development.

With this in mind, the vast majority of available computation power and wind tunnel testing is dedicated to external flow simulations and the aerodynamic development of the car with the majority of the internal flow simulations and thermal analysis being performed at the beginning of the season and during the initial development phase of the race car with minimal investigations completed in-season. This has two main disadvantages with regard to optimal design of the cooling strategy.

The first disadvantage is that the thermal simulations, on which the cooling strategy is designed, often has to be run with preliminary boundary conditions as specifications often change throughout the development phase with respect to the final design

that will be used on track. The second disadvantage is that the power unit is also allowed to be developed throughout this season. This obviously means that these preliminary boundary conditions on which the cooling strategy is predicated may also change throughout the season. Hence, a safety factor must be included within these preliminary boundary conditions to account for any increases in thermal loading that may occur as a side effect of the race car and power unit development throughout the season.

Both of these points add uncertainty to the boundary conditions on which the thermal simulations are run, and on which the thermal management strategy is predicated. As mentioned previously, reliability is key to success in Formula One, of which the thermal management strategy is a major factor. Hence, in order to factor in the aforementioned uncertainty in thermal boundary conditions, over design of the thermal management strategy is inevitable in order to ensure the reliability of the final race car design.

1.3 Use of Experiments in Formula One

1.3.1 Development of race car aerodynamics

In the design process, it is possible to define three different categories of fidelity for information that is used to inform the design process. These are, in ascending order of fidelity (and expense), computational simulations, wind tunnel testing and on-track testing. As such, the design methodology in Formula One has traditionally been heavily reliant on experimental data, with most of the top teams making heavy use of aerodynamic wind tunnel testing. This was particularly the case for teams that had proprietary access to wind tunnel testing facilities where experiments were often conducted non stop throughout night and day. The majority of teams do not have access to their own race track to conduct on-track testing, thus making on-track testing economically infeasible for component development purposes.

In recent times, the use of experimental data has declined due to both improvements in computational simulations and the FIA Governing Body's restrictions (see Section 1.2.2). On-track testing of modern race cars (deemed as those from the previous two seasons) is also limited to a set number of weekends per year [4]. However, it is

widely accepted that the sole use of computational simulations to inform the design process without experimental validation has limited value. As such, most teams retain an experimental program that is designed to validate computational simulations (as opposed to being focused on the development of the race car design).

1.3.2 Use of experiments in power unit development

The design of the power unit is not subject to the aforementioned restrictions on computational simulations and experimental testing as the power unit manufacturers are considered separate entities to the racing teams. This is necessary as, at the time of writing, there are only four power unit manufacturers which means that the majority of the teams are supplied by external manufacturers. The key factor in determining the performance of the power train is the efficient extraction of work from the combustion gases throughout the engine cycle. As such, the development of the power unit is heavily dependent on experimental data, given the difficulty in accurately simulating the complex multi-phase, multi-species combustion reactions that occur within the cylinder.

Engine manufacturers often have access to a large set of experimental facilities that will feature both whole system and sub-system breakdowns of the power unit on individual test benches. Development is often undertaken on a component by component basis, but whole system testing often comprises a large proportion of the testing program. This is crucial given that both the performance and reliability of the power unit is heavily dependent on successful system wide integration of all the constituent parts of the power train.

Each team is limited to a set number of power units that can be used in each season (four of each of the six sub-components considered to constitute a full power unit; internal combustion engine, motor generator unit - kinetic, motor generator unit - heat, energy store, turbocharger, and control electronics). Any additional components used above this limit are subject to racing penalties (in the form of starting grid demotions). This restriction requires power unit manufacturers to consolidate their upgrades into discrete packages (as opposed to the incremental evolution method prevalent in aerodynamic development) which can then be implemented when a power unit component is changed.

Whilst upgrades can still be made to the power unit throughout the season, the current system means that the number of upgrades throughout the season is dependent on the reliability of the power unit (as replacements for failed components are also counted in the aforementioned limits). Thus, it is essential that each power unit component is extensively tested for reliability (in addition to performance) before it is implemented on the race car.

1.4 Computational Analysis in Formula One

As described in Section 1.2.2, the FIA Governing Body has placed restrictions on the wind tunnel testing and computational simulations that can be run by each team as part of a package of cost cutting measures. In order to extract the maximum amount of useful information within these restrictions, the use of computational analyses in Formula One is becoming increasingly important for in-season development with experimental data mainly being used as validation for computational results. As such, the design process and subsequent evolution of the race car design throughout the season is incredibly reliant on the use of computational simulations.

The restrictions on computational simulations mainly apply to computational fluid dynamics (CFD) simulations which are used to simulate the external flow both over and under the race car and the internal flow through the underhood domain. The former are used to design and characterise the aerodynamic performance of the race car by simulating the airflow over various aerodynamic surfaces, while the latter are normally used in order to inform the thermal management strategy and cooling system in order to ensure the reliability of the components in the underhood area.

These restrictions on computational power and wind tunnel testing has led to a reduction in the amount of raw data available that can be processed and fed in to the design process. This scarcity of raw data has placed a new emphasis on the concept of data efficiency, whereby it is necessary to extract the maximum information possible from the raw data that can then be used by engineers to inform choices during the design process.

1.5 Aims of Current Thesis

The aim of this research project is to investigate various methods of analysing and managing the thermal loads in the underhood region and power unit of a Formula One race car. The various methods outlined in this thesis originated from a collaboration between the research team (author and principle investigator) and industry partner. As such, from this discussion, three main areas of research are presented in this thesis:

1. A study of methods for economising on restricted simulations and maximising the amount of useful information that can be extracted from the raw data produced by each computational simulation
 - With restrictions on computational power requiring teams to economise on the computational simulations they run, it is becoming increasingly important to either extract the maximum amount of useful information from the simulations they run, or to develop alternative methods to restricted simulations in order to generate the same data.
 - This led to the development of an efficient computational model that would allow the design team to generate temperature field predictions whilst also economising on the amount of computational simulations that they run. The computational model was designed such that it was outside the activities restricted by the FIA Governing Body and should provide a similar level of accuracy to results produced by equivalent restricted computational analyses.
2. A novel method for measuring in-cylinder heat fluxes during engine running
 - The measurement of in-cylinder surface temperatures and heat fluxes is difficult due to the harsh environment inside the cylinder of an internal combustion engine whilst the engine is running. As such, limited experimental data is available to validate existing empirical heat transfer models for modern engines. This is particularly the case for high performance engines such as those used in Formula One.

- In addition to being able to survive in the aforementioned harsh environments, the rotational speed of the crank shaft requires the sensor to record data at a high bandwidth. The aim of this research was to develop a sensor that could measure crank angle resolved temperatures for a Formula One engine, which is limited to 15,000 RPM. This necessitates a sensor response frequency of 100 kHz.
- This required the development of a novel type of sensor that would be able to fit into existing fittings for instrumentation on a single cylinder test bench, without additional modifications. The sensor should be able to survive the environment inside the cylinder of an internal combustion engine during operation and be able to measure temperature on a crank angle resolved time scale.

3. Prediction of cylinder wall temperature using empirical data

- Although the measurement of the cylinder wall surface temperature and heat fluxes are useful to gauge the raw performance of the cylinder design, the information can also be used to improve the power unit performance in other ways.
- The aim was to develop a computationally efficient code that could use data from the previous combustion cycle in order to provide real-time information on the cylinder state that can be used to optimise the combustion phase of the following engine cycle.

1.6 Research Contributions

1. A new linear superposition method has been developed that is able to predict the forced convection dominated temperature field in complex geometries from a fixed number of datum computational analyses. The model is flexible in terms of the geometry, number of heat sources in the system and the thermal boundary conditions applied to the system. The resulting temperature field predictions have shown to be of comparable accuracy to computational simulations with the same explicit thermal boundary conditions with negligible differences.

2. The linear superposition method was then extended to not only support varying thermal boundary conditions, but also varying inlet boundary conditions. The linearity of the system breaks down as the inlet boundary conditions move away from the inlet boundary conditions in the datum analyses. In order to account for this, a non-linear correction is proposed, based on the domain being analogous to a circular ducted heat exchanger.
3. A low speed wind tunnel was designed and commissioned in order to validate the aforementioned linear superposition method. The test section was designed to replicate the quarter scale rear-end model of a 2014 Formula One car as used in the computational model. The internal assembly was fitted with electrical heater pads in order to replicate heat sources in the computational domain and was instrumented with thermocouples, thermochromic liquid crystals and pressure tappings in order to validate the computational model. A novel method of combining individual images using image processing techniques was investigated and implemented.
4. A novel thin film gauge sensor was developed in order to measure the surface temperature inside the cylinder of an internal combustion engine. The sensor was designed to use an existing instrumentation mount, avoiding the need for significant modifications to the existing test facility. Using post-processing techniques, this allows calculation of the surface heat flux. The rise time of the sensor was calculated and shown to be compatible with the target response frequency of 100 kHz. In order to survive the harsh conditions within the cylinder, a platinum thin film gauge was deposited on a ceramic substrate that was mounted in a 3-D printed stainless steel collet.
5. The aforementioned sensor was tested on a single cylinder, air cooled test bench in order to measure the surface heat flux inside the cylinder. The experimental data was compared using the Woschni [5] and Hohenberg [6] empirical heat transfer models. An amorphous CVD diamond coating was trialled in order to increase the robustness of the sensor.
6. A computationally efficient method of converting heat flux into in-cylinder wall surface temperature predictions using fast Fourier transforms and finite differ-

encing has been demonstrated. The method is designed to predict in-cylinder wall surface temperatures during engine running which is difficult to measure during engine operation and also factors in the influence of the cylinder cooling jacket on the external surface of the cylinder wall. The proposed method is designed to run on the engine control unit (ECU) with minimal processing time in order to include the influence of the cylinder wall surface temperature when calculating the spark advance of the next cycle.

1.7 Overview of Thesis Structure

This thesis is divided into the following chapters:

Chapter 2 summarises previous research into the analysis and management of the thermal loads that occur in automotive vehicles. Given that Formula One teams tend to avoid public disclosure of their design process, this summary is fairly generic in terms of the applications considered (both commercial and consumer vehicles are examined). Both internal flow and combustion related activities are examined in line with the topics addressed in this thesis. Previous research into high bandwidth and high heat flux instrumentation is also examined, with applications from both the aerospace and automotive industries considered.

Chapter 3 introduces the linear superposition theory and explains the mathematical principles behind the proposed method and the required information to use the model. The implementation of the linear superposition model in MATLAB is also described and the approaches to account for varying thermal and kinematic boundary conditions are detailed.

Chapter 4 describes the results of using the linear superposition method applied to a geometrically simplified quarter scale model of a Formula One race car using computational simulations. The results are then compared to computational simulations with the same explicit boundary conditions. Results for both varying thermal and kinematic boundary conditions are shown. An alternative geometry is also included for additional validation of the linear superposition method. The improvements in computational efficiency compared to full computational simulations are also shown.

Chapter 5 shows the experimental validation of the linear superposition model (detailed in Chapter 3) using an experimental facility. Thermocouple rake measurements are used to provide spatial temperature measurements in the outlet plane of the model. Thermochromic liquid crystals are used to gain spatial distributions of heat transfer coefficient on internal surfaces and cross sectional temperature profiles using a mesh suspended in the gas flow. Data is collected using an array of borescopes, with the resulting separate image streams combined to provide a larger field of view.

Chapter 6 studies the development of novel thin film gauge technology designed to measure surface temperatures in hostile environments, in particular inside the cylinder of an internal combustion engine. The response frequency of the sensor is specified to be 100 kHz in order to capture crank angle resolved surface temperature measurements at 15,000 RPM.

Chapter 7 presents results from the sensor developed using the technology described in Chapter 6. The thin film gauges are placed inside a single cylinder, air cooled test bench in order to measure the surface temperature, which is then post-processed to calculate heat flux. This experimental data is then compared to the Woschni [5] and Hohenberg [6] empirical heat transfer models.

Chapter 8 describes the development of a model to calculate the inside surface temperature of a cylinder in an internal combustion engine from knowledge of the heat flux signature using fast Fourier transforms and finite differencing.

Chapter 9 shows the implementation of the aforementioned computational model in MATLAB and compares the output of the model to experimental data. The improvements in computational efficiency compared to finite element models are also shown.

Chapter 10 aims to summarise the findings of the thesis and also includes some recommendations for future work.

1.8 Publications

Lim, Ireland, & Collett (2016) [7] details the development of a linear superposition method in order to rapidly predict the temperature field in complex geometries

from an initial set of computational analyses used to characterise the domain. The method is demonstrated on a geometrically simplified quarter scale model of a Formula One race car conforming to the 2014 FIA Governing Body regulations, although the method is applicable to all complex geometries.

The linear superposition method allows accurate predictions of the temperature field within a complex geometry for varying thermal boundary conditions with negligible computational cost (compared to full computational simulations). Different parts within the geometry are used as heat sources to simulate the real world thermal boundary conditions. The method is shown to give results that show a similar degree of accuracy to computational simulations with the same explicit boundary conditions. A non-linear correction is also presented to allow the model to account for changes in inlet velocity to the system.

This work was presented at the SAE Thermal Management Systems Symposium in Mesa, Arizona, USA in October 2016.

Lim, Ireland, & Collett (2017) [8] is a more complete description of the presentation by Lim, Ireland, & Collett [7] and includes some preliminary experimental work which is described in more detail in this thesis. This paper has undergone peer-review and has been accepted for publication as a technical paper as part of the SAE World Congress 2017. The paper is currently being considered for publication as part of the SAE Mechanical Systems journal.

This paper has been presented at the SAE World Congress in Detroit, Michigan, USA in April 2017

Chapter 2

Literature Review

2.1 Introduction to Literature Review

Given that the amount of literature in the public domain surrounding research and development in Formula One is very small, this literature review will include a more generic review of the literature surrounding the core topics described in this body of research. The benefits of publishing in academic journals is not considered a sufficient incentive for most Formula One teams given that this disclosure of results and innovation has the potential to erode competitive advantage. As such, most teams avoid public disclosure, even if it offers legal protection (for example, patenting), in order to maintain their competitive advantage.

This literature predominantly draws on literature examples from the consumer and industrial vehicle sectors for the automotive specific topics. The majority of literature on the subject of high temperature, high frequency instrumentation is associated with the aerospace industry as there are a number of similarities to gas turbines and hypersonic applications in terms of sensor robustness requirements, frequency response requirements and measurement environment.

2.2 Thermal Analyses of the Underhood Region

2.2.1 The importance of characterising underhood thermal behaviour

Thermal analysis in the underhood region presents numerous issues to automotive design engineers for a variety of reasons. The placement and packaging of the powertrain components in the underhood region creates a geometrically complex environment that will contain a multitude of heat producing components which must be adequately cooled to ensure they remain within a certain temperature range, allowing them to function optimally and reliably. In addition to this, modern design trends have also driven a demand for tighter packaging of the power train components for both aerodynamic and functionality gain which increases the geometrical complexity of the environment. This combination of complex geometry, and resultant flow field complexity, coupled with the requirement to understand the thermal environment within the underhood region creates unique challenges for automotive designers.

The importance of characterising the thermal behaviour of the underhood environment is twofold. Firstly, as previously mentioned, the underhood region tends to contain the majority of the powertrain components which are vital to the operation of the vehicle. The performance of these components are temperature sensitive and so it is vital that there is an adequate provision of coolant to keep the components within a certain temperature range so as to allow them to operate as required. This is compounded by the fact that many of these components will produce their own heat, which must be accounted for in the thermal management strategy. Taking this to the extreme, an adequate provision of coolant must be supplied such that the components do not fail, which may either cause the vehicle to fail, or worse, put the safety of the vehicle passengers at risk.

While the thermal management strategy must provide adequate air flow to the radiators and components in the underhood region to achieve sufficient cooling, the over-provision of this coolant air flow has related performance penalties. The provision of coolant generally results in an associated aerodynamic penalty caused by the the pressure drop of the airflow through the radiators (also known as radiator drag).

Hence, as modern manufacturers strive to increase the fuel economy of their vehicles, the over provision of coolant must be avoided to achieve the maximum efficiency of the vehicle [9].

Striking the optimal balance between reliability and efficiency is the key to the successful design of the thermal management system which requires a characterisation of the thermal behaviour of the system. In order to acquire this understanding of the system thermal behaviour, it is important to investigate the characteristics of the system at various points throughout the expected operating range of the vehicle [10].

2.2.2 Current design methodology for the underhood environment

The geometrical complexity of the underhood region makes it difficult to fully characterise the thermal behaviour of the system using experimental investigations [11]. Hence, the design process is heavily reliant on computational analyses to predict and characterise the thermal behaviour of the underhood region at a variety of operating points, which in turn, informs both the thermal management strategy for the components located in the underhood region and the operating life of the system components. These hurdles, combined with commercial pressures to reduce the design time scales and the subsequent time to market, has meant that a computationally efficient and accurate iterative numerical approach is becoming increasingly crucial to the design process of a successful modern vehicle.

The internal flow field within the underhood region is often forced convection dominant as the internal flow through the underhood region is driven by the dynamic head of the vehicle or by fans located on the radiator assembly [12]. An example of the internal flow complexity is shown in Figure 2.1 which shows the streamlines through the underhood geometry of an industrial heavy truck.

As noted above, experimental studies of the underhood region are generally in short supply due to the complexity in instrumenting the area sufficiently to glean any useful information. Studies by Khaled, Harambat, and Peerhossaini [14] showed, using experimental techniques, that it is possible to decouple and separately evaluate the different heat transfer mechanisms within the region. This approach is often

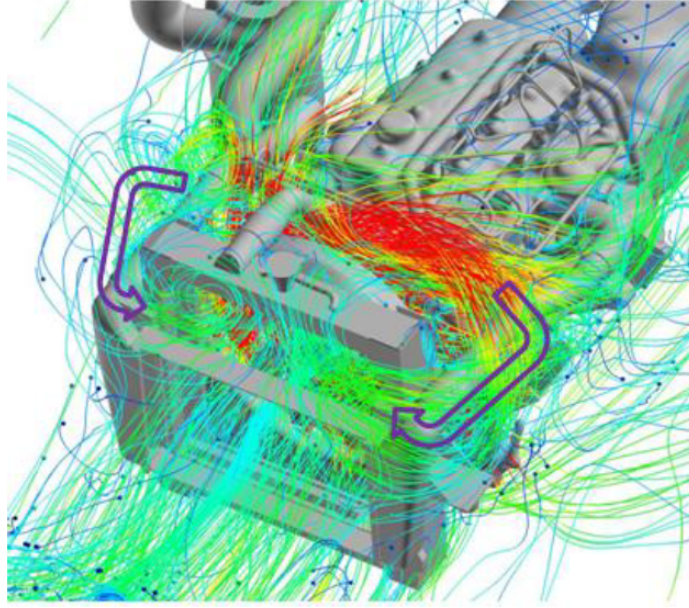


Figure 2.1: Example of underhood flow field streamlines [13]

mirrored in computational analyses as different mesh fidelities are required to predict the effects of each heat transfer mechanism [15].

Khaled, Harambat, and Peerhossaini [14] comment that the agreement between 1-D computational methods and experimental point temperature measurements was approximately within 3.7% on average. The authors also apply a similar iterative approach to investigate the thermal behaviour of the system at a variety of operating points. Limited experimental validation of 3-D computational codes using point temperature measurements is reported by Fortunato et al. [11]. This is a common trend within the industry whereby experimental point measurements tend to be used for validation of computational techniques, rather than being used for design investigations. This is due to the difficulty of extensively instrumenting a representative domain without altering the original flow field.

As a surface mesh and solid mesh is required to computationally calculate the effects of radiative and conductive heat transfer, it is often the convective heat transfer calculation which acts as the bottleneck for the computational simulations. With this in mind, the different heat transfer mechanisms are often decoupled and solved separately. The convective heat transfer within the underhood region is highly dependent on the complex internal flow field within the region resulting in studies of convective

heat transfer being computationally expensive. As the effective use of CFD is always subject to a balance between a desire for both speed and accuracy, a computational analysis of the underhood region is often simplified at the expense of accuracy, to ensure that an analysis can be completed within a feasible amount of time. Figure 2.2 shows an example of the geometry simplification applied to the internal powertrain assembly in a 3-D underhood simulations as reported by Fortunato et al. [11].

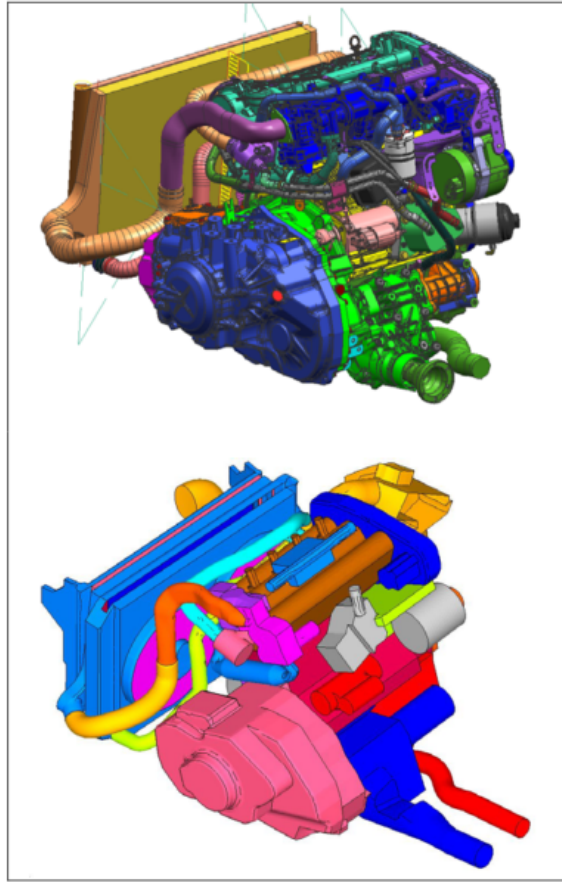


Figure 2.2: Example of geometry simplification of the powertrain for underhood simulations [11]

Bayraktar [16] reports the example of a 1-D simulation of a military ground vehicle underhood zone that can be completed in under 2 hours on a desktop machine. This is compared to a geometrically simplified 3-D CFD simulation taking over 24 hours to run on a HPC cluster. Bayraktar [16] asserts that, while 1-D simulations tools provide a valuable first prediction about a proposed system, full characterisation of the system thermal behaviour is still likely to require higher fidelity 3-D CFD simulations. An

accuracy comparison between 1-D simulation tools and 3-D simulations tools are provided by Mao, Feng, and Michaelides [17]. The authors also note the difficulty in achieving convergence of a full 3-D simulation which can partially be attributed to the difficulty in creating a high quality mesh due to the geometrical complexity of the domain.

Additional simulations of the underhood region of a passenger vehicle have been performed by Ljungskog and Nilsson [18]. A CFD analysis of the underhood internal flow was performed using the commercial STAR-CCM+ CFD package. The time required for 3000 iterations (considered to be sufficient for a converged steady solution) using 160 CPU cores on a HPC cluster was reported to be approximately 7-8 hours. Given the complexity of the internal flow field within the underhood region, the authors noted that careful choices of solver parameters are required to achieve a converged result within a reasonable amount of time.

It is common practice to run 3-D simulations of the underhood environment on HPC clusters in order to reduce the solution time. An example of this was a 3-D simulation of a detailed underhood geometry for a Land Rover LR3 consumer vehicle, reported by Alajbegovic, Sengupta, and Jansen [19]. The analysis was performed on a 64 CPU HPC cluster and was run for 30,000 iterations (considered to be sufficient for the steady state solution to reach convergence). An additional 1-D calculation was used to generate the boundary conditions of the internal surfaces which vary according to the performance of the cooling system. The entire simulation time was reported to be 66 hours.

2.3 Use of Linear Superposition for Thermal Analyses

Under some conditions, the differential equation for temperature in a flow field (or a conducting body) can be linear in the temperature variable. This means that solutions for complex thermal boundary conditions can be built up by superposing the solution for similar cases.

The use of linear superposition to predict a resultant temperature field has previously been investigated in the power electronics industry. A similar iterative investigation

procedure is employed (similar to the automotive design process mentioned above) whereby characterisation of the temperature field at multiple operating points is imperative to ensure the longevity of any given design. The results of this investigation can then be used to inform the thermal management strategy for the system [20].

Gauche [21] sets out a method for using linear superposition as a complimentary tool to a traditionally wholly CFD based design methodology in order to maximise the information that can be extracted from data generated using CFD. In the power electronics sector, the use of linear superposition allows the engineer to create a flexible thermal model early on in the design process. This thermal model can be created with preliminary thermal boundary conditions in the absence of component choices or experimental data. This thermal model can then be updated (as opposed to having to recreate the entire model) when more accurate and experimentally validated thermal boundary conditions for each components become available.

By varying the model boundary conditions, rather than redefining the entire model, the use of linear superposition lends itself to acceleration of an iterative investigation. The creation of a parametric model from existing CFD data is discussed by Gauche and Road [22]. An example of superposition, as applied to multi-chip power electronic modules is outlined by Sofia [23]. The thermal behaviour of a multi-chip package is described using linear expressions for the thermal resistance of each chip. These single heat source solutions can then be superposed to generate solutions for problems with more complex thermal boundary conditions.

The limitations of this method, as discussed by Gauche and Wei [24], are evident in scenarios where there is a mixture of heat transfer mechanisms that are relevant in the system. The authors note that the effects of natural convection and radiation are important when considering the thermal behaviour of passively cooled printed circuit boards. As these mechanisms change the governing differential equation for temperature from linear, the linear superposition method cannot be applied. Gauche and Wei suggest that the effects of natural convection and radiation can be partially captured by linearising the relationship between heat dissipated in the chip and the resultant temperature difference. The linearisation is applied using a perturbation calculated

gradient value and closest datum point (generated using computational simulations). It is noted that calculation of the gradient requires additional computational analyses.

Zahn [25] also notes the problems caused by radiative and natural convective heat transfer when applying the linear superposition techniques. The author discusses the use of a non-linear corrective factor, used after the linear superposition technique has been applied, which is calculated by testing the device over the entire range of power dissipations expected. Zahn notes that this correction is not required for forced convective cooling environments.

2.4 Investigations of In-Cylinder Thermal Conditions

2.4.1 Introduction

In internal combustion (IC) engines, any heat loss from the combustion gases to the cylinder wall and coolant is an important factor in establishing engine cycle efficiency. The heat loss from the combustion gases during the engine cycle constitutes a large proportion of the energy released during the combustion phase. This has a large effect on the work that is extracted from the engine cycle, and thus, the overall engine performance and efficiency. An in depth discussion on the effects of IC engine heat transfer on overall performance is discussed by Borman and Nishiwaki [26].

In recent years, increasing emphasis has been placed on improving the efficiency of IC engines in order to meet increasingly demanding environmental regulations. A significant factor in improving IC engine efficiency is the requirement of accurate predictions for the thermal state of the system throughout the engine cycle, most notably the in-cylinder condition. Improving the accuracy of these thermal predictions allows for many potential benefits, such as improving the cooling system, reducing friction caused by thermal expansion mismatches, and more accurate simulation in the design process through improvements in boundary condition accuracy.

When investigating in-cylinder heat transfer, the three mechanisms of heat transfer must be considered. Radiative heat transfer is a significant factor when considering heat transfer in compression ignition diesel engines due to the formation of soot

particles during combustion which are able to radiate significant heat [27]. These dark soot particles act as black bodies which increases the amount of radiative heat transfer. However, for spark ignition engines (such as those used in Formula One), radiative heat transfer is much less significant. Therefore, forced convection is investigated when examining the heat transfer between the combustion gases and the cylinder wall, whilst conduction is considered when investigating the heat transfer through the cylinder wall.

The heat transfer that occurs in IC engines is a spatially non-uniform and highly unsteady phenomena with heat fluxes that can oscillate between near-zero and up to 5MW/m^2 (depending on the engine specification) throughout the cycle. Negative heat transfer can occur during the intake stroke as the induced air at the start of the intake phase is cooler than the cylinder wall. A thorough summary of the empirical correlations for in-cylinder heat transfer has been collated by Finol and Robinson [28].

2.4.2 Empirical models to predict in-cylinder heat transfer

Empirical 0-D models have traditionally been used to predict the heat transfer inside the cylinder of an IC engine due to the difficulty in both instrumenting the area to provide empirical data and in accurately simulating the complex flow field physics inside the cylinder during the engine cycle.

Empirical models have traditionally been based on correlations for Nusselt number, derived from experimental investigations of forced convectional heat transfer for turbulent flow inside circular tubes [29]. Dimensional analysis can be used to show that the relevant non-dimensional groups are the Nusselt (Nu), Reynolds (Re), and Prandtl (Pr) numbers. These are typically in the power law form shown in Equation 2.1.

$$Nu = aRe^bPr^c \quad (2.1)$$

The values of the non-dimensional coefficients (denoted a, b, c) are determined using regression analysis from empirical data. Changes in the Prandtl number can often

be neglected due to the negligible changes in working fluid properties for gases in the operating regime, and so the canonical form of the empirical model is often solely based on Reynolds number (as in Equation 2.2).

$$Nu = aRe^b \quad (2.2)$$

Equation 2.2 is used to relate the cylinder heat transfer coefficient h (contained in Nu) to the gas properties (k_g , ρ_g , μ_g), characteristic velocity, and length of the cylinder. Mean piston speed V_p and cylinder bore B are often used to define the characteristic velocity and length of the cylinder. Calculation of the heat transfer coefficient then allows the calculation of the heat flux into the cylinder wall caused by forced convection using the definition of the heat transfer coefficient shown in Equation 2.3.

$$h = \frac{q}{(T_g - T_w)} \quad (2.3)$$

Annand [30] proposed the first widely used empirical correlation for Nusselt number, shown in Equation 2.4. The cylinder gas Reynolds number is defined in Equation 2.5.

$$q = \left(\frac{k_g}{B} \right) aRe_g^b (T_g - T_w) + c(T_g^4 - T_w^4) \quad (2.4)$$

$$Re_g = \frac{\rho_g v_{p,m} B}{\mu_g} \quad (2.5)$$

Equation 2.4 contains a supplementary term to account for the radiative heat transfer between the bulk gas and cylinder. This equation was based on a review of experimental data from two compression ignition engines (one being a two stroke and the other being a four stroke engine). The cylinder bulk gas temperature was calculated using the cylinder pressure, charge air mass, and cylinder volume via the ideal gas law.

Annand [30] found that the b and c coefficients did not vary between the two engines,

but the a coefficient (which scales the convective heat transfer) showed large variation between the two engines. Thus, the a coefficient is generally considered to be a scaling factor depending on engine geometry. Suggested parameters for the a , b , and c coefficients are summarised and reported by Watson and Janota [31].

Watson and Janota [31] also noted that the use of the mean piston velocity as the characteristic velocity to determine the bulk gas Reynolds number is not entirely accurate and that there are other parameters, other than piston velocity, that influence the bulk gas velocity. While the empirical model described in Equation 2.4 was initially proposed using experimental data derived from compression ignition engines, recent experimental work has suggested alternative values for the a , b , and c coefficients that can be used to adequately describe spark ignition engines behaviour [31].

An updated and improved empirical version of Equation 2.4 was proposed by Annand and Ma [32]. The updated model is designed to take into account the spatial and temporal variation in the cylinder wall heat flux. This is shown in Equation 2.6.

$$q = \left(\frac{k_g}{B} \right) Re_g^{0.7} \left[a(T_g - T_w) - \frac{b}{\omega_c} \frac{dT_g}{dt} \right] + c(T_g^4 - T_w^4) \quad (2.6)$$

The initial Annand empirical model (see Equation 2.4) is based on a quasi-steady approach which approximates the heat transfer rate at any given time point as proportional to the difference in temperature between the bulk gas and the cylinder wall.

The updated model is designed to take into account the phase lag between the temperature difference and heat flux into the cylinder wall, caused by the thermal capacity of the working fluid. This is accomplished with the addition of a temperature derivative term.

In order to account for the spatial variation in heat flux, the model coefficients were calculated using temperature measurements from a variety of different locations on the cylinder head. Typical values for the a , b , and c coefficients are reported by Annand and Ma [32]. It should be noted that coefficient c in Equations 2.4 and 2.6

is not dimensionless and is often expressed as a multiple of the Stefan-Boltzmann constant ($\sigma = 56.7 \times 10^{-9} \text{ W m}^{-2} \text{ K}^{-4}$).

Another widely used empirical correlation for in-cylinder heat transfer was proposed by Woschni [5]. The model is also based on the standard form for forced convection of turbulent flows in circular tubes (given as Equation 2.2). The Woschni correlation is presented in Equation 2.7. The Woschni model relates the instantaneous conditions to a reference condition in the engine cycle (such as the opening of the intake valve).

The Woschni model is applied separately to different phases of the engine cycle to represent the varying heat fluxes throughout the cycle. The model contains two coefficients C_1 and C_2 which vary depending on the phase of the engine cycle. This approach is intended to account for the variation of the gas velocity throughout the engine cycle. The C_0 term is used to scale the correlation and is normally calibrated during motored operation of the engine under test.

Typical values for the C_1 and C_2 coefficients are shown in Table 2.1. The data in Table 2.1 shows that the C_2 term is only considered during the combustion phase of the engine cycle. This is because the C_2 term accounts for the additional kinetic energy of the cylinder gas caused by the energy released during the combustion process, on top of the kinetic energy imparted by the piston velocity.

$$h = C_0 p^{0.8} B^{-0.2} T_g^{-0.53} \times \left[C_1 v_p + C_2 \frac{V_s T_r}{p_r V_r} (p - p_m) \right]^{0.8} \quad (2.7)$$

Table 2.1: Typical values for Woschni empirical model [5]

Engine Cycle Phase	C_1	$C_2 (\frac{\text{m}}{\text{sK}})$
Gas Exchange (Intake & Exhaust)	6.18	0
Compression	2.28	0
Combustion & Expansion	2.28	3.24×10^{-3}

A disadvantage of using this correlation, as outlined by Finol [28], is the necessity to evaluate the motored pressure of the engine. Watson and Janota [31] suggest an energy equation based model for accurately computing this pressure, but an alternative simpler method of calculating the motored pressure using the known reference

conditions and an assumption of isentropic compression and expansion was shown to be reasonably accurate.

The third commonly used empirical model for in-cylinder heat transfer was proposed by Hohenberg [6] and is shown in Equation 2.8. The empirical model proposed by Hohenberg follows a similar derivation to earlier empirical models in that it is based on experimental data and previous models (in this case the Woschni model outlined in Equation 2.7).

The major difference between the Hohenberg model and the models outlined previously is the definition of the characteristic length, based on the diameter of a sphere whose volume corresponds with the cylinder volume at a given point in time (as opposed to cylinder bore). This allows the characteristic length (and consequently, the Reynolds number) to vary with crank angle. As before the C_0 term is used to scale the empirical correlation with the heat transfer at the motored condition. A common value for the C_1 term is 1.4 m/s.

$$h = C_0 V^{-0.06} p^{0.8} T_g^{-0.4} (v_p + C_1)^{0.8} \quad (2.8)$$

Experimental data collected by Hohenberg [6] showed that the Woschni empirical model tended to underestimate the heat flux during the compression and exhaust phases and overestimated the heat flux during the combustion phase. A comparison of the empirical models can be found in Figure 2.3.

2.4.3 Validation of empirical heat transfer models

As described in Section 2.4.2, all of the empirical models presented above were developed from experimental data. As such, these empirical models all contain coefficients that can be tuned to enable the model to fit a set of given experimental data. While this approach can be considered to be useful to validate the models, it is less useful when it comes to producing thermal predictions during the design phase of an engine. This is particularly the case for engines which are significantly different to the engines used in the derivations. While three models have been presented in Section

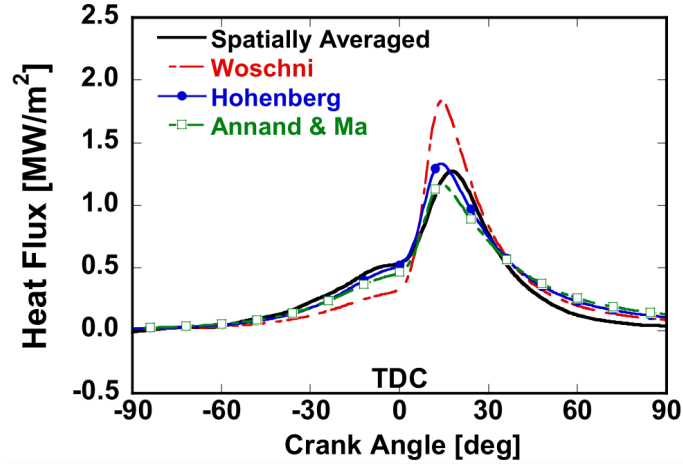


Figure 2.3: Comparison of empirical heat transfer models [33]

2.4.2, there are a multitude of other approaches where the coefficients of each model have been tuned to allow a better fit with a given set of experimental data.

Wimmer [34] reports that the application of each of the empirical heat transfer models described above each gives very different results depending on engine type, load, and speed which makes a relative assessment of their merits difficult.

2.4.3.1 Experimental validation

With this in mind, the experimental validation of these empirical models is clearly very important. The flow field inside the cylinder represents a very harsh environment whereby the gas pressure, gas temperature and the mixing of fuel and air during the engine cycle present a very harsh and unsteady environment that is difficult to measure. As the peak gas temperature can reach up to 2500 K, active cooling of the cylinder head, cylinder block and piston is often required to manage the thermal loading.

Surface temperature is typically the quantity measured in order to determine the heat flux inside the cylinder. The heat flux from the in-cylinder gas to the cylinder wall can then be calculated from the surface wall surface temperature which must satisfy the Fourier equation (shown in Equation 2.9). A comprehensive early review of inverse methods used to determine heat flux from surface temperature is given in Schultz and Jones [35].

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \quad (2.9)$$

Traditionally, experimental studies of in-cylinder heat transfer have been carried out using fast response or eroding type thermocouples at point locations throughout the cylinder assembly. However, temperature measurements using thermocouples are typically limited to a maximum response frequency of 10 kHz which, depending on the rotational speed of the engine, may not be sufficient to provide crank angle resolved accuracy. In order to increase the bandwidth of the thermocouples, the junction between the two elements must be reduced in size which tends to increase the fragility of the thermocouple, making them unsuitable for IC engine applications, where the environment is particularly aggressive.

A report summarising the transient response of the varying thermocouple technologies was produced by Gatowski, Smith, and Alkidas [36]. A summary of the different temperature measurement techniques used in IC engine research is shown below.

- **Thin Film Gauges**

Thin film gauges use the variation in resistance of a thin metal film in order to measure temperature. One advantage of using thin film gauges is the increased bandwidth as the low mass of the thin film can allow a frequency response of greater than 100 kHz which makes them better suited to fully capture the unsteady features of the combustion process. Another advantage is that the thin film gauges are also considered to be more durable and robust than fast response thermocouples (depending on the system used). Thin film gauges were first used for hypersonic research [37] and gas turbine applications [38]. Recently, they have been adapted for use in IC engine research by Wilson et al. [39] and Demuynck [40].

The disadvantage of using thin film gauges is that particulates from the combustion process deposited on the gauge can affect the measurement accuracy. Another disadvantage is the difficulty in routing the signal from the thin film gauge out from the high temperature environment (although the challenges of cabling and connections are common to all sensors).

- **Coaxial Thermocouples**

Coaxial thermocouples were developed by Bendersky [41] and were the first type of thermocouple typically used IC engine heat transfer research. Coaxial thermocouples consist of a single thermocouple element, coated with insulation that is then put into a tube which forms the second thermocouple element (shown in Figure 2.4a). The end of the probe is then plated with a thin layer of metal to form the electrical connection. Examples of measured heat flux using coaxial thermocouples is reported by Sihling and Woschni [42] and Overbye et al. [43].

- **Pair-Wire Thermocouples**

The pair-wire thermocouple consists of the two thermocouple elements (insulated with a ceramic oxide layer) inserted into a metal tube (an example is shown in Figure 2.4b). The top surface is then ground down to form a flush surface, where the junction between the thermocouple elements is then created by the deposition of a metal layer. The metal layer is often vacuum deposited to reduce the thickness which increases the response time of the thermocouples. Pair-wire thermocouples of this type were used by Hohenberg [6] and Wimmer et al. [34] for IC engine studies.

One issue with this type of thermocouple, as reported by Wimmer [34], is the difficulty in maintaining the one-dimensional condition required to solve for heat flux using the Fourier equation (see Equation 2.9).

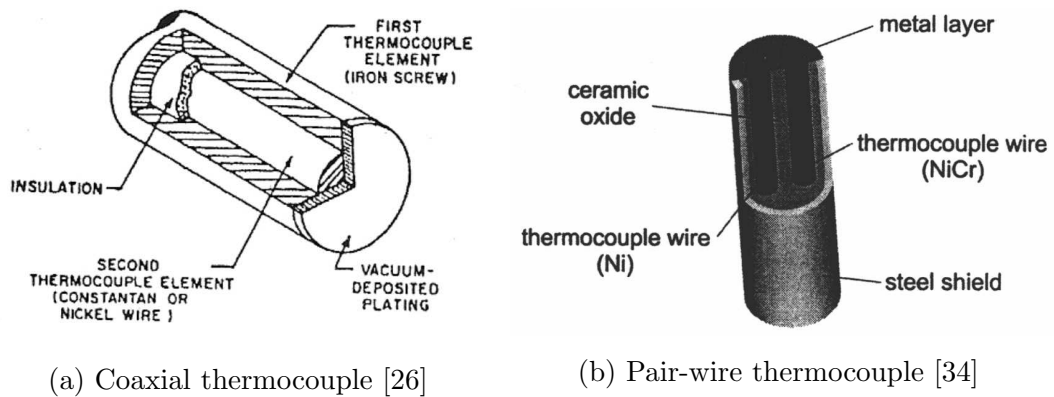


Figure 2.4: Overview of IC engine temperature measurement techniques

- **Eroding Ribbon Thermocouples**

The eroding ribbon thermocouple consists of the two thermocouple elements manufactured as thin sheets, separated by a thin mica layer that insulates the two elements from each other (see Figure 2.5) as described by Buttsworth [44]. A micro-junction is formed across the two elements by drawing an abrasive surface or sharp implement across the junction. This creates small metallic elements that bridge the insulation at the surface. For use in IC engine experiments, the junction will often be renewed by small particles in the combustion gases eroding the surface [40]. Eroding ribbon thermocouples have been used by Alkidas [45], Stone et al. [46], Wang et al. [47], [48], Buttsworth et al. [49], and Rakopoulos and Mavropoulos [50] for combustion engine applications.

Eroding ribbon thermocouples are considered to have the fastest response of all the thermocouple options. However, finite element analysis studies have shown that lateral conduction effects can cause inaccuracies in the calculation of surface heat flux from the resultant surface temperature measurements for certain timescales [51].

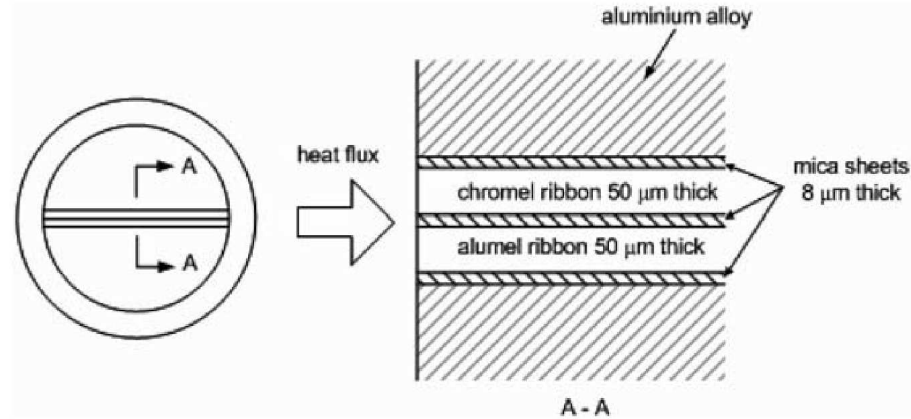


Figure 2.5: Eroding ribbon thermocouple [52]

2.4.3.2 Computational validation

The highest level of fidelity for computational simulations of combustion engine heat transfer is a full 3-D simulation of the in-cylinder gas throughout the engine cycle.

The use of computational fluid dynamics (CFD) simulations allow for a detailed investigation of the heat transfer mechanisms that occur during the engine cycle. Full 3-D simulations are often used in conjunction with the simpler 0-D models described in Section 2.4.2, such as in Mohammadi and Yaghoubi [53].

The simulation of the flow field throughout the engine cycle is difficult due to the multi-phase and multi-species nature of the flow and the mixture of convective and conduction heat transfer.

An approach to solve for the simultaneous heat transfer mechanisms was reported by Liu and Reitz [54] that utilised two separate programs (a CFD solver for the fluid flow and a FEA solver for heat conduction in the solid). This decoupled approach used the heat flux into the cylinder wall from the CFD program as a thermal boundary condition for the FEA program. The updated surface temperature was then passed back into the CFD program as an updated boundary condition for the next iteration in the CFD program. This approach showed better agreement with experimental data, but was reported to be computationally expensive due to the large volumes of data transfer required between the two separate solvers.

An alternative method is the use of a conjugate heat transfer model. The conjugate heat transfer model is a coupled approach to solve for the fluid domain and solid domain simultaneously. The advantage of this approach is that the conjugate heat transfer method uses the same numerical methods to solve for both the fluid and solid domains, which allows for accurate prediction of the temperature at the fluid-solid interface. The conjugate approach is utilised by Xin et al. [55] to evaluate the metal temperatures surrounding the combustion chamber. Li and Kong [56] also uses the conjugate method to simulate the combustion process throughout the engine cycle. Both publications report the advantages of using the coupled approach to better capture the unsteady, non-uniform temperature distributions on the combustion chamber surfaces.

2.4.4 Calculating heat release from pressure measurements

The temperature distribution of the cylinder wall is heavily dependent on the heat release during the combustion phase of the engine cycle. Peak heat transfer occurs

due to the compression of the gases by the piston and the energy released during the burning of the air fuel mixture. Although engine manufacturers aim for the combustion process to be as repeatable and predictable as possible, cyclic variation is common due to the stochastic features caused by the ratio, distribution, and turbulent motion of the air fuel mixture in the cylinder and the resulting flame propagation [57].

Pressure is considered to be the best known quantity to analyse the combustion process [58]. Although in-cylinder pressure is a difficult quantity to measure accurately in a cost-effective manner, some high performance engines (such as those used in motorsport) do use transducers to measure in-cylinder pressure during engine operation. For engines without direct pressure measurement, various methods of reconstructing the in-cylinder pressure from other quantities such as engine speed and loading have been reported in literature [59], [60].

The first reported method to ascertain heat release from pressure data was described by Gatowski [61] and is based on the first law of thermodynamics (neglecting any changes in mass and specific gas constant).

$$dU = dQ - dW \quad (2.10)$$

Rearrangement of Equation 2.10 and use of the ideal gas relationships above leads to Equation 2.11. Research by Brunt, Rai, and Emtage [62] has shown that the the accuracy of heat release predictions made using Equation 2.11 is vastly improved by factoring in the temperature dependency of the specific heat capacity ratio, γ .

$$dQ = \left(\frac{\gamma(T)}{\gamma(T) - 1} \right) p dV + \left(\frac{1}{\gamma(T) - 1} \right) V dp \quad (2.11)$$

Equation 2.11 can then be integrated with respect to the crank angle to estimate the net heat release during the engine cycle using in-cylinder pressure. Whilst a signification portion of this energy will be released as exhaust enthalpy, a large portion will also be transferred to the cylinder walls which must also be accounted for.

2.5 Spark Advance Optimisation

2.5.1 Introduction

Another key area of research in the field of spark ignition internal combustion engines is the study of spark advance control. The correct spark advance timing is crucial for both IC engine efficiency and power as it controls the initiation of the combustion process and the evolution of the cylinder pressure as the piston moves and extracts work. If the spark advance is too early, potential work is lost to heat transfer and the compression process (as the pressure rises whilst the piston is moving upwards) while if the spark advance is too late, potential expansion work is lost [63]. This is illustrated in Figure 2.6 which shows the engine brake torque produced by a spark ignition engine as the spark advance angle is varied.

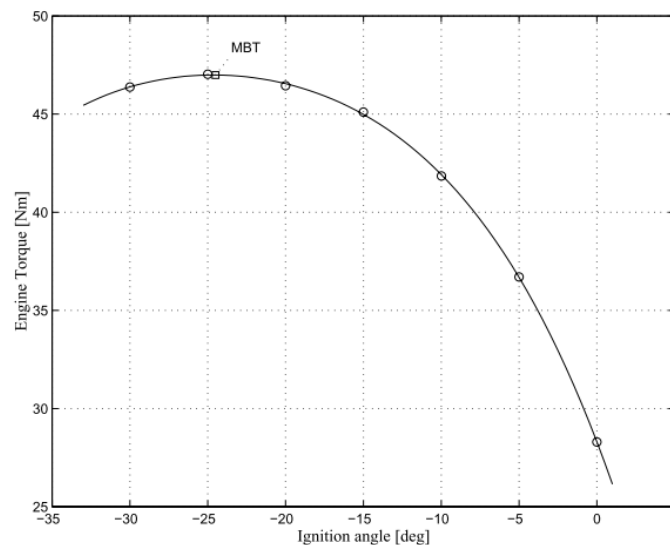


Figure 2.6: Engine brake torque as a function of spark ignition angle. Position of maximum brake torque (MBT) is marked [63]

The theoretical optimal spark advance can be defined as the timing that maximises engine efficiency as this results in maximum production of work. The spark advance initiates the combustion of the air-fuel mixture of the cylinder and development of the in-cylinder pressure with relation to the piston position which is key to the work extracted. However, it should be noted that the spark advance used during engine op-

eration may differ from the optimal value in order to prevent uncontrolled combustion events (such as knock), which could cause damage to the engine.

The optimal spark advance is highly dependent on the engine geometry and thermal management strategy as the in-cylinder pressure development is influenced by a number of factors, such as the flame propagation through the combustion chamber, heat transfer to the walls, and pressure losses through piston blowby. This means that the optimal spark advance timing is highly specific to each individual engine design, as well as the operating point of the engine.

The flame propagation through the cylinder is not only dependent on the geometry, but a range of operational characteristics such as engine speed, engine loading, cylinder wall temperature, intake air temperature, fuel quality, and the air-fuel ratio. However, many of these are difficult to measure in a real-time, cost-effective manner. Therefore, the approach of most manufacturers is to use a subset of these parameters which are easy to measure in order to estimate the spark advance for each cycle. Parameters such as engine speed, coolant temperature, and intake air temperature are relatively accessible and economical to measure and so are often used to calculate spark advance [63].

2.5.2 Spark advance control

In modern consumer spark ignition engines, the spark advance timing is controlled via an open loop control system (meaning there is no feedback from the combustion process to the spark advance calculation on a cycle by cycle basis) implemented using the engine control unit (ECU). Open loop control is used in consumer vehicles due to the cost of robustly instrumenting the engine cylinder, which is required to provide a feedback signal. A disadvantage of using open loop control is that the calibrated spark advance timings are often conservative to account for the possible variation in the parameters that are not measured in order to ensure adequate performance, and thus can be sub-optimal where these unmeasured parameters do not change. This conservatism can be mitigated, to some degree, using closed loop control providing the feedback signal used provides enough information about the combustion process [64].

Eriksson [65] describes a method of using ion-current control as another method of providing the feedback signal necessary to implement a closed loop method of calculating spark advance. It was noted that whilst ion-current control provides a cost effective method of implementing closed loop control, the ion-current signal is post processed to extract information on pressure development. Therefore, the preferred method of implementing feedback is to use a transducer [66] to monitor the in-cylinder pressure development as the work output of the engine cycle is calculated using Equation 2.12. However, the expense of fitting pressure transducers able to endure the combustion process makes this method economically impractical for production vehicles. In order to avoid damage to the engine caused by knock events, production vehicles use accelerometers in order to detect engine cycles suffering from knock. When afflicted cycles are detected, the spark advance is then retarded in order to move the engine away from the knocking regime.

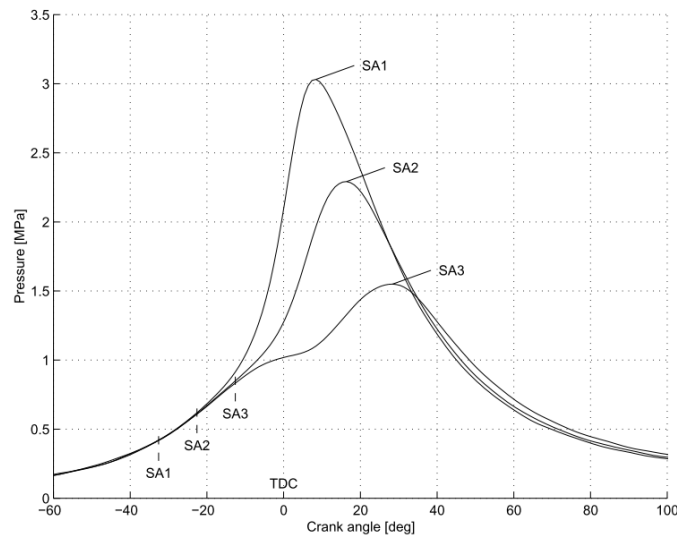


Figure 2.7: Pressure distribution in the cylinder for three different spark advance timings [63]

Figure 2.7 presents the pressure distribution in the cylinder for different spark advance timings. Although the highest pressure occurs for the earliest spark advance timing, this does not produce the maximum work output as this is dependent on the pressure peak location relative to the change in volume (e.g. relative to the direction of piston movement) as in Equation 2.12.

$$W_c = \oint_{cycle} p(\theta) dV \quad (2.12)$$

2.5.3 Calculation of spark advance values

Heywood [67] described a method of producing data for look up tables used to implement the open-loop control system for determining spark advance for spark ignition engines using an engine dynamometer. Firstly, the spark advance timing corresponding to the production of maximum brake torque (MBT) is located. The spark advance timing is then retarded (towards TDC) until the engine brake torque is approximately 1% below MBT.

The calibrated spark advance timings are calculated across the IC engine's operating range which has been characterised on the dynamometer. These values are then stored in look up tables such that they can be utilised for normal engine operation. Other approaches are used to calculate spark advance timings for operating conditions that are not measured on the dynamometer such as engine idle and cold start conditions.

2.5.4 Retardation of spark advance and engine knock

The spark advance timing is often retarded from the MBT point for two reasons [63]. Firstly, retarding the spark advance lowers the mean temperature of the combustion process which can lead to significant reductions in the the NO_x emissions produced during the engine cycle. Secondly, retarding the spark advance decreases the likelihood of the in-cylinder conditions reaching engine knock conditions by reducing peak pressure. Increasing the margin between the operating condition and the knock-producing condition allows the lifetime of the cylinders to be maximised as knock can cause damage to the engine. Damage associated with engine knock is generally caused by increases in peak heat transfer to the cylinder walls. Work by Lu et al. [68] showed that peak heat flux increases linearly with peak pressure for engine knock events, thus increasing the likelihood of damage.

Engines that operate with knock generally suffer from low cyclic efficiency due to the increased heat loss to the cylinder walls. However, intermittent knock is not necessarily indicative of poor spark advance timing as engine knock is considered a

stochastic event and it is possible that other engine cycles not affected by knock may achieve a higher work output due to the increased combustion speed [69]. Therefore, the spark advance should be chosen such that the frequency of knock afflicted cycles does not significantly impact on the longevity of the engine. As running closer to knock-producing condition does increase the work output of the engine cycle (at the expense of engine lifespan), the margin between the operating condition and knock-producing condition is often smaller for race cars than for consumer road vehicles.

Retardation of the spark advance from the optimal timing will inevitably result in a reduction of work extracted from the cycle. However, as the engine brake torque is relatively insensitive to small changes in the spark advance at the MBT value (see Figure 2.6), the associated work penalty is considered to be insignificant compared to the aforementioned advantages.

2.6 Conclusions

The literature reviewed in this section covers a wide range of heat transfer research in the automotive industry.

Thorough thermal characterisation of the underhood region presents a challenging problem to design engineers, owing to the geometrical complexity and associated computational expense of simulating the flow in the domain. This leads to a required balance between simulation fidelity and the number of cases in the operating window that can be investigated [11] [19]. The use of linear superposition for thermal analysis presents a possible solution to the aforementioned problem, but has previously been limited to much simpler domain geometries in the power electronics industry [24].

The field of in-cylinder heat transfer research is well established with recent developments in experimental measurements of the heat flux into the cylinder wall during the engine cycle. Eroding thermocouples have successfully been used to measure the in-cylinder heat transfer during the engine cycle [47]. The use of thin film gauges for in-cylinder heat transfer experiments is fairly immature, with scope for faster response times at the expense of sensor robustness [70]. The importance of experimentally measuring the in-cylinder heat transfer is illustrated by the variation in predicted values calculated using empirical models, such as the Woschni [5] and Hohenberg [6] models.

Accurate measurements and predictions of the in-cylinder heat transfer are vital to the successful design of an efficient and robust spark ignition combustion engine. The in-cylinder heat transfer is one of the factors in selecting the optimal spark advance to extract maximum work from the engine cycle without suffering from engine knock [63]. Due to the cost of robustly instrumenting the engine cylinder, look up tables are often used to select the spark advance for a given engine condition based on a set of measurements at various accessible locations in the powertrain [67]. However, the spark advance values contained in the look up tables are often conservative in order to account for variance in the non-measured parameters, such as the cylinder wall temperature (which can be a potential cause of engine knock). This conservatism can potentially lead to a reduction in engine cycle efficiency when the non-measured parameters do not change.

Chapter 3

Linear Superposition Theory

3.1 Introduction

The linear superposition of solutions from simplified (or component) boundary conditions in order to predict a resultant temperature field for more complex boundary conditions has previously been investigated within the power electronics industry in order to establish the cooling requirements of printed circuit boards for convective dominated environments [21], [20], [24]. Applications of this method have been restricted to limited locations within the domain (such as integrated circuit chips and particular solder joint temperatures) and primarily two dimensional cases. The aim of the present work is to explore the feasibility of extending the use of linear superposition to a complex three dimensional domain, such as the environmental temperatures under the hood, for which heat transfer is dominated by forced convection.

3.2 Linear Superposition of the Convection Transfer Equation

For an incompressible flow field (in which the density, thermal conductivity and specific heat capacity of the working fluid can be regarded as sensibly independent of temperature), the fluid temperature at any point, $T(x, y, z)$, must satisfy the partial differential equation (PDE) for forced convection, defined in Equation 3.1, which is linear in temperature [29].

$$u \frac{\delta T}{\delta x} + v \frac{\delta T}{\delta y} + w \frac{\delta T}{\delta z} = \frac{k}{\rho c_p} \left[\frac{d^2 T}{dx^2} + \frac{d^2 T}{dy^2} + \frac{d^2 T}{dz^2} \right] \quad (3.1)$$

As a linear and homogeneous PDE, it is possible to use the superposition principle (or linearity principle) to state that if T_1 and T_2 are solutions to Equation 3.1, then $T = c_1 T_1 + c_2 T_2$ is also a solution to Equation 3.1 with any constants c_1 and c_2 [71].

This superposition principle allows new solutions of a given linear, homogeneous PDE to be obtained from given solutions by adding them or multiplying them by any constants (or a combination of the two). In the case of Equation 3.1, superposition allows a solution of the fluid temperature for a complex set of boundary conditions to be built from a scaled combination of individual component solutions.

This is shown in Figure 3.1, where the temperature at a point in a simple convecting field for a set of thermal boundary conditions can be calculated using the superposition of two individual component solutions.

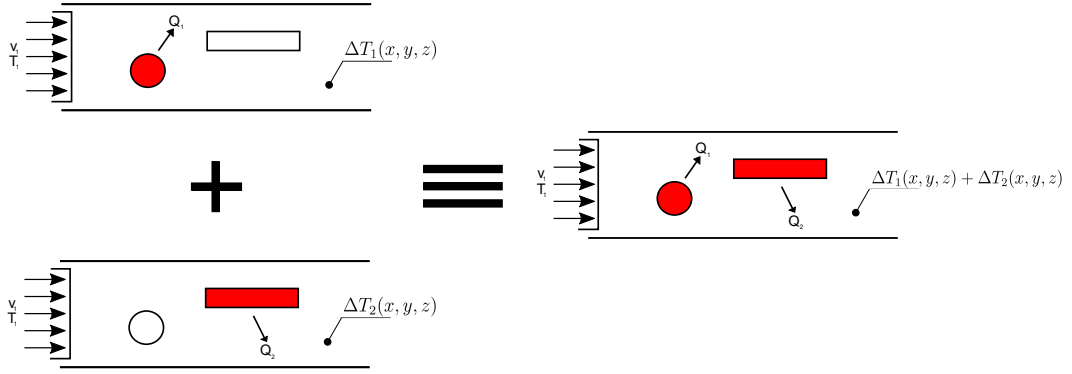


Figure 3.1: Superposition principle as applied to a convective heat transfer problem

3.3 Heat Transfer Dimensional Analysis

It is possible to apply dimensional analysis to investigate the relationship between the temperature field and the thermal boundary conditions in the system. The interrelated physical quantities chosen for dimensional analysis were density (ρ), velocity (u),

characteristic length (D), dynamic viscosity (μ), specific heat capacity (c_p), thermal conductivity (k), temperature change (ΔT), and heat flow (Q).

Table 3.1 lists the physical quantities in the system and their relevant physical dimensions. With 8 physical quantities and 5 fundamental dimensions, it is possible to use Buckingham's Pi theorem to identify 3 related non-dimensional groups.

Table 3.1: Exponents for the repeating variables used in the dimensions of the interrelated variables

		Variables							
		ρ	u	D	μ	c_p	k	ΔT	Q
Dimensions	M (kg)	1	0	0	1	-1	0	0	0
	L (m)	-3	1	1	-1	0	-1	0	0
	T (s)	0	-1	0	-1	0	-1	0	-1
	θ (K)	0	0	0	0	-1	-1	1	0
	H (J)	0	0	0	0	1	1	0	1

Using ρ , u , D , ΔT , and Q as repeating variables to express the 5 physical dimensions, 3 non-dimensional groups of physical quantities were formed using the remaining variables. After some manipulation, the following non-dimensional groups are obtained:

- Π_1 Reynolds number - $\frac{\rho u D}{\mu}$
- Π_2 Prandtl number - $\frac{c_p \mu}{k}$
- Π_3 Dimensionless Temperature Change - $\frac{\Delta T}{\frac{Q}{\dot{m} c_p}}$

Application of Buckingham's Pi theorem, allows us to express one of these groups as a function of the remaining groups as in Equation 3.2.

$$\Pi_3 = f(\Pi_1, \Pi_2) \rightarrow \frac{\Delta T}{\frac{Q}{\dot{m} c_p}} = f(Re, Pr) \quad (3.2)$$

From the above dimensional analysis, it can be seen that the dimensionless temperature change within the system can be expressed as a function of the remaining dimensionless groups, the Reynolds and Prandtl numbers. If the working fluid is kept

constant, then it is possible to ignore Prandtl number in situations where variation in the properties of the working fluid, caused by temperature change, are negligible.

Similarly, in situations when the geometry and flow speed are also constant, it is possible to disregard the system dependency on Reynolds number. Using the same assumptions (that changes in the working fluid are negligible and that the flow speed is constant) we can reduce the dimensionless temperature change group to a linear relationship between the heat flow rate into the system and the temperature change of the working fluid.

For heat flow into the system, it is necessary to look at the heat transfer occurring at the domain boundary. Taking the case of an individual thermal boundary condition, labelled j , at which the total heat flow into the surface of the domain is Q_j , it is possible to write the following equation to represent the boundary condition (see Equation 3.3).

$$-k \frac{\delta T}{\delta y} = \frac{Q_j}{A_j} \quad (3.3)$$

The y dimension is defined as perpendicular to the surface and A_j is the area of the surface through which the heat flows. As Equation 3.3 is linear in both heat and temperature, this allows linear scaling of the thermal boundary conditions.

If multiple heat flows are now applied simultaneously (see Figure 3.1), the solution of the temperature field in the air flow, $T(x, y, z)$, can be obtained by superposing scaled solutions for temperature with individual heat flows applied independently. For each independent thermal boundary condition case, the wall boundary condition elsewhere is set to be adiabatic.

3.4 Implementation of the Linear Superposition Method

3.4.1 Definition of the temperature influence matrix

The implementation of linear superposition to analyse convective temperature fields is aided by the definition of a temperature influence coefficient, ψ (Equation 3.4) using

the linear relationship defined in Section 3.3. Each temperature influence coefficient prescribes the predicted temperature rise per unit heat flow at a given point i in the domain for a certain heat source j , such that:

$$\psi_{i,j} = \frac{\Delta T_i}{Q_j} \quad (3.4)$$

Hence, for a given domain with m nodes and n discrete heat sources, an m by n matrix of temperature influence coefficients can be defined as follows:

$$\overline{\psi}_{mn} = \begin{bmatrix} \psi_{1,1} & \psi_{1,2} & \dots & \dots & \psi_{1,n} \\ \psi_{2,1} & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots \\ \psi_{m,1} & \dots & \dots & \dots & \dots \end{bmatrix} \quad (3.5)$$

Equation 3.5 is termed the *temperature influence matrix* and is a collation of all the temperature influence coefficients for each node and heat source in the domain. Thus, once the values of the temperature influence matrix are known, the temperature rise at any point in the domain can be calculated for any combination of applied heat loads using Equation 3.6.

$$\overrightarrow{\Delta T}_m = \overline{\psi}_{mn} \overrightarrow{Q}_n \quad (3.6)$$

3.4.2 Determination of the temperature influence matrix

One way to populate the temperature influence matrix is to analyse the thermal field generated by each discrete source individually. This is done by isolating each thermal boundary condition in turn and analysing the thermal field generated whilst the isolated thermal boundary condition, i , is set to a known, pre-defined heat flux with all the other thermal boundary conditions in the system set to zero.

$$\begin{bmatrix} \psi_{1,i} \\ \vdots \\ \psi_{m,i} \end{bmatrix} = \frac{1}{Q_i} \begin{bmatrix} \Delta T_1 \\ \vdots \\ \Delta T_m \end{bmatrix} \quad (3.7)$$

This reduces the temperature influence matrix in Equation 3.6 to a one dimensional vector that can be easily solved to find the temperature influence coefficients relating to each node in the domain for the individual heat source (as in Equation 3.7). These one dimensional vectors can then be concatenated together to form the temperature influence matrix as in Equation 3.5. Once the temperature influence matrix has been populated, it is then possible to use Equation 3.6 to predict the temperature rise that would be produced with each heat source set to a certain heat flow.

3.4.3 Discretisation of continuous domains

A key step in the implementation of the linear superposition method is the division of the continuous domain into discrete elements and nodes. The use of computational analyses (such as CFD) already necessitates the partitioning of continuous volumes into discrete elements whose properties can be considered to be homogeneous throughout. The resulting discretised domain is known as a mesh and can either be considered to be structured or unstructured depending on whether the connectivity of the individual elements is related to their storage and numbering arrangement.

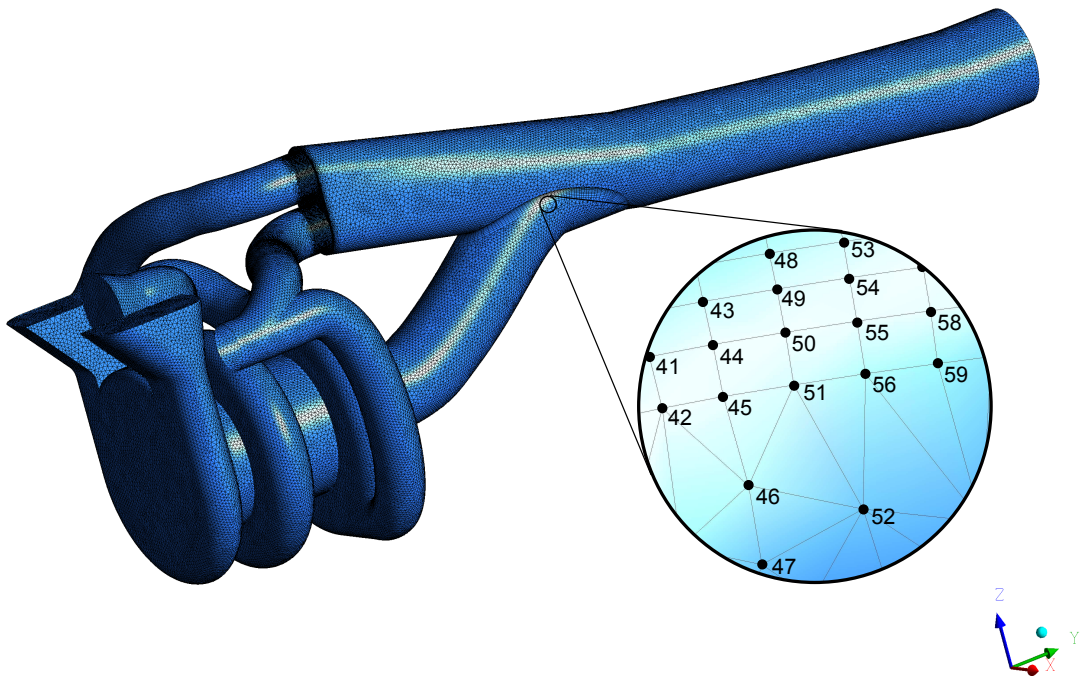


Figure 3.2: Discretised exhaust turbocharger assembly

The linear superposition method is a complementary tool to CFD analyses and utilises the same discretisation approach. The values output from a computational mesh can either be cell centered values or node centred values. Figure 3.2 shows the discretisation of a turbocharger and exhaust assembly using unstructured mesh notation (where each element is given a unique number) with node centered values considered. If the geometry under consideration remains identical between analyses, each unique identifier will continue to represent the same element in 3-D space for each computational analysis which is essential for accurate implementation of the linear superposition method.

3.5 An Approach to Account for Varying Flow Velocities

By using the dimensional analysis conducted in Section 3.3, it is possible to adapt Equation 3.6 in an approach that would allow the model to take into account changes in the inlet velocity boundary conditions. The dimensional analysis yielded three non-dimensional groups (shown below), with dimensionless temperature change being expressed as a function of the remaining dimensionless groups, the Reynolds and Prandtl numbers.

- Π_1 Reynolds number - $\frac{\rho u D}{\mu}$
- Π_2 Prandtl number - $\frac{c_p \mu}{k}$
- Π_3 Dimensionless Temperature Change - $\frac{\Delta T}{\frac{Q}{\dot{m} c_p}}$

In Section 3.3, an assumption was made that changes in the working fluid physical properties were negligible, allowing the Prandtl number dependency to be ignored.

With the Prandtl number dependency considered negligible, Buckingham's Pi theorem can then be used to express the dimensionless temperature change within the system as a sole function of Reynolds number. As the dimensional analysis cannot provide the actual form of the relationship between dimensionless temperature change and Reynolds number, it is necessary to draw on empirical data and correlations to assess the extent of the Reynolds number dependency.

If the domain under consideration has a single inlet plane and a single outlet plane and is fully enclosed, we can draw an analogy between the system and a simple pipe flow problem, allowing the system to be analysed as a circular duct heat exchanger. In the case of a circular duct heat exchanger with a low rate of heat transfer, it is possible to show that the temperature change observed in the domain is a weak function of Reynolds number. This relationship is explored in Section 3.6.

If the Reynolds number dependency on dimensionless temperature change is firstly neglected, then, for a given set of thermal boundary conditions, we would expect a linear relationship between the temperature change and mass flow through the system. In fact, the steady flow energy equation can be used to show that the relationship between heat and change in mixed bulk temperature is independent of Reynolds number. Equation 3.8 follows directly from the steady flow energy equation, where ΔT denotes the change in gas bulk temperature.

$$Q = \dot{m}(c_p \Delta T) \quad (3.8)$$

Equation 3.8 can be re-arranged as in Equation 3.9.

$$\frac{\Delta T}{Q} = \frac{1}{\dot{m}c_p} \quad (3.9)$$

There are advantages to re-expressing the temperature influence coefficients as dimensionless values (shown in Equation 3.10).

$$\psi_{i,j} = \frac{1}{\dot{m}c_p} \xi_{i,j} \quad (3.10)$$

Replacing the temperature influence coefficients, ψ , in Equation 3.6 with the new non-dimensional temperature influence coefficients, ξ , leads to the definition of a new temperature influence matrix, as in Equation 3.11. This dimensionless temperature influence matrix can be populated using the same method as outlined in Section 3.5.

$$\frac{1}{\dot{m}c_p} \begin{bmatrix} \xi_{1,1} & \xi_{1,2} & \dots & \dots & \xi_{1,n} \\ \xi_{2,1} & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots \\ \xi_{m,1} & \dots & \dots & \dots & \dots \end{bmatrix} \vec{Q}_n = \overline{\Delta T}_m \quad (3.11)$$

3.6 Application of a NTU- ϵ Correction Factor

As introduced in Section 3.5, the flow in the underhood system can be considered similar to the flow in a closed pipe (with single inlet and outlet planes) as it is enclosed without a free surface. By then drawing an analogy to a duct heat exchanger with a small number of transfer units, it was possible to assume that the dependency of the dimensionless temperature change group on Reynolds number was negligible. The assumption of poor heat transfer was considered valid as the primary design aim of the system was not to maximise heat transfer.

By utilising empirical relationships from the same heat exchanger analysis, it is possible to estimate the form of the relationship between these the groups identified during the dimensional analysis. This allows the effect of the Reynolds number dependency on the system to be taken into account.

3.6.1 Heat exchanger analysis

By considering the system as a heat exchanger, the convective efficiency of the system can be defined as the ratio of heat transfer that occurs in the system to the theoretical maximum heat transfer that could be achieved. For systems where the thermal properties of the working fluid are uniform, this is calculated using the maximum temperature difference in the system (see Equation 3.12).

$$\eta_c = \frac{Q}{Q_{max}} = \frac{T - T_{in}}{T_{max} - T_{in}} \quad (3.12)$$

It is then possible to express the convective efficiency in terms of the non dimensional Stanton number [72]. This is shown in Equation 3.13.

$$\eta_c = 1 - \exp(-4.St.\frac{L}{D}) = 1 - \exp(-NTU) \quad (3.13)$$

For systems which are not primarily designed for heat transfer purposes, it is possible to assume that the number of transfer units (NTU) in the system will be small. Hence, it is then possible to expand the exponential term in Equation 3.13 using a Taylor series (as in Equation 3.14). If the second order terms and above are discarded, it is possible to approximate the convective efficiency of the system (η_c) using the number of transfer units in the system.

$$\eta_c = 1 - [1 - NTU + \frac{(NTU)^2}{2!} - \dots] \approx NTU \quad (3.14)$$

From the approximation in Equation 3.14, it is then possible to approximate the convective efficiency at each point in the domain using the number of transfer units (see Equation 3.15).

$$\eta_c = \frac{T_i - T_{in}}{T_{max} - T_{in}} \approx NTU \quad (3.15)$$

As the number of transfer units can be expressed in terms of Stanton number (see Equation 3.13), this can then be substituted into Equation 3.15 in order to express the convective efficiency at a given node in the domain using Stanton number, as in Equation 3.16.

$$\frac{T_i - T_{in}}{T_{max} - T_{in}} \approx -4.St.\frac{L}{D} \quad (3.16)$$

If both the geometry and thermal boundary conditions in the system remain constant, this enables us to reduce Equation 3.16 to a linear relationship between the temperature rise and Stanton number at a given point i (see Equation 3.17). The coefficient $C_{1,i}$ is some constant that captures the thermal boundary conditions (Q_{max}) and characteristic dimensions ($\frac{L}{D}$) at each point in the domain, which do not change between each simulation as the thermal boundary conditions and geometry are both locked at this point.

$$C_{1,i}(T_i - T_{in}) = St_i \quad (3.17)$$

3.6.2 Stanton number as a function of Reynolds number

Equation 3.17 describes a linear relationship between the temperature rise at each point in the domain and the Stanton number at that point. The non-dimensional Stanton number can be expressed as a function of dimensionless Reynolds, Nusselt and Prandtl numbers as shown below in Equation 3.18:

$$St = \frac{Nu}{Re Pr} \quad (3.18)$$

The empirical Dittus-Boelter correlation for fully developed, turbulent pipe flow is then used to approximate the Nusselt number using the Reynolds and Prandtl numbers (shown in Equation 3.19) [73].

$$Nu = 0.023 Re^{0.8} Pr^{0.4} \quad (3.19)$$

If the changes in the working fluid physical properties are ignored, this allows the Prandtl number dependency to be disregarded. This reduces the Stanton number to be a sole function of Reynolds number. If the same assumption of constant working fluid physical properties is applied to Reynolds number, this then allows us to express the Stanton number solely in terms of the flow velocity at a given point i , as in Equation 3.20, where C_2 and C_3 are constants containing the working fluid properties.

$$St = C_{2,i}(Re^{-0.2}) = C_{3,i}(u^{-0.2}) \quad (3.20)$$

Using the relationships outlined in Equations 3.20 and 3.17, it is then possible to define the form of the relationship between the temperature rise and Reynolds number outlined using Buckingham's Pi theorem in Section 3.3.

3.7 Linear Superposition using Constant Temperature Boundary Conditions

The proposed method, described in detail above and summarised in Equation 3.6, sets out a method of predicting the temperature rise at a given point in the domain for a set of heat flow boundary conditions. However, it can be advantageous to use constant temperature boundary conditions to define the problem rather than heat flows.

For a system where forced convection is the only significant heat transfer process, heat flux (q) can be calculated from the convective heat transfer coefficient (h) and the difference between the surface temperature (T_s) and gas temperature (T_g) as in Equation 3.21.

$$h = \frac{q}{(T_g - T_w)} \quad (3.21)$$

Equation 3.21 defines a linear relationship between the heat flux emitted and a temperature difference. In many cases, the convective heat transfer coefficient is solely dependent on aerodynamics and does not vary with thermal boundary conditions [74]. This allows the heat flux from a given surface to scale with temperature difference.

Using Equation 3.21, it is possible to replace the heat flux boundary conditions with a temperature difference term. To ensure the correct scaling of the temperature field with the constant temperature boundary conditions, it is logical to define a reference temperature (T_r) which is kept consistent throughout the analysis, in order to calculate the temperature difference.

When using temperature difference rather than heat flux in the linear superposition technique, the temperature influence coefficients described in Equation 3.6 become non-dimensional. As such, it is clearer to describe these using a new designator, ζ , so as not to confuse them with the non-dimensional temperature influence coefficients described in the mass flow scaled method (see Equation 3.11).

Therefore, for a system with n thermal boundary conditions and m points in the domain, the linear superposition method can be described using Equation 3.22, where

$T_{w,n}$ describes the constant temperature applied to the boundary condition denoted n .

$$\begin{bmatrix} \zeta_{1,1} & \zeta_{1,2} & \dots & \dots & \zeta_{1,n} \\ \zeta_{2,1} & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots \\ \dots & \dots & \dots & \dots & \dots \\ \zeta_{m,1} & \dots & \dots & \dots & \dots \end{bmatrix} \left(\begin{bmatrix} T_{w,1} \\ \vdots \\ T_{w,n} \end{bmatrix} - \vec{T}_r \right) = \vec{\Delta T}_m \quad (3.22)$$

3.8 Temperature Dependent Non-Linear Effects

3.8.1 Changes in working fluid properties

The linear superposition method (as outlined in Section 3.3) assumes that the working fluid physical properties will not change throughout the analysis. This allows the Reynolds and Prandtl number dependencies (identified using Buckingham's Pi theorem) to be disregarded when scaling with respect to the thermal boundary conditions, and the Prandtl number dependency to be disregarded when scaling the temperature field with inlet velocity.

However, if the temperature field predicted using the linear superposition method is vastly different from the temperature field observed during the initial characterisation analyses, it is probable that several of the temperature dependent material properties will vary from the initial analysis. This invalidates the assumption that the Reynolds and Prandtl number dependencies can be disregarded.

3.8.2 Natural convection effects

As stated in Section 3.2, the linear superposition method assumes that the temperature field in the domain is forced convection dominant. The key dimensionless group to determine valid use of the linear superposition method is the Grashof number (shown in Equation 3.23).

$$Gr_D = \frac{g\beta(T_s - T_\infty)D^3}{v^2} \quad (3.23)$$

The Grashof number describes the ratio of buoyancy to viscous forces in the working fluid. Therefore, the ratio of Grashof number to Reynolds number (as shown below), can be used to determine the effects of natural convectional heat transfer in the domain. As the effect of natural convectional heat transfer is non-linear, it cannot be predicted using the linear superposition method and as such, invalidates the use of the technique described in Section 3.5.

$$\begin{aligned}\frac{Gr}{Re^2} &>> 1 && \text{Natural convection dominant} \\ \frac{Gr}{Re^2} &\approx 1 && \text{Mixed natural and forced convection} \\ \frac{Gr}{Re^2} &<< 1 && \text{Forced convection dominant}\end{aligned}$$

As described in Section 3.8.1, certain properties of the working fluid are likely to be temperature dependent. The key physical quantity to determining the effects of natural convection is the density gradient within the domain. As density is a temperature dependent property for most working fluids, natural convection effects can occur where high density gradients exist due to large variations in the temperature field.

If these large temperature gradients do not exist in the initial characterisation analyses used to populate the temperature influence matrix, any buoyancy induced natural convection effects, caused by changes in working fluid density, will not be captured in the subsequent analyses.

As described above, an assumption is made that the system is forced convection dominant. For application to the underhood domain of a Formula One race car travelling at 120 kph with a temperature difference of 400 K, the Reynolds number is $\sim 8.8 \times 10^5$, whilst the Grashof number is $\sim 2.8 \times 10^9$. This gives the ratio of $\frac{Gr}{Re^2}$ as 276 which, using the criterion above, indicates that the system is forced convection dominant and that the effects of natural convection can effectively be disregarded.

Chapter 4

Linear Superposition Computational Results

4.1 Computational Model

4.1.1 Computational mesh

In order to determine the coefficients in the temperature influence matrix, it was necessary to run an initial set of CFD analyses. In each of these CFD analyses, a hybrid tetrahedral-prism mesh, comprising approximately 5.6 million cells, was generated using ANSYS ICEM.

The domain under consideration consisted of a quarter scale rear bodywork model of a 2014 Formula One race car, derived from a public domain CAD model¹. Inside the rear bodywork geometry is a geometrically simplified model of a Formula One power train. Five surfaces on the internal assembly (shown in Figure 4.1) were used to define non-adiabatic thermal boundary conditions. These surfaces were chosen, in consultation with colleagues at our industrial partner, to represent significant sources of heat in the underhood environment at the following locations:

1. Turbocharger
2. Left Cam Cover
3. Right Cam Cover

¹Public domain CAD sourced from www.exchange3D.com

4. Exhaust

5. Gearbox

The thermal boundary conditions at all other model wall surfaces were set to adiabatic. The two side pod inlets were used as the inlet planes with the flow exiting the domain through the single outlet plane at the rear of the model. The inlet and outlet planes are highlighted in red in Figure 4.1.

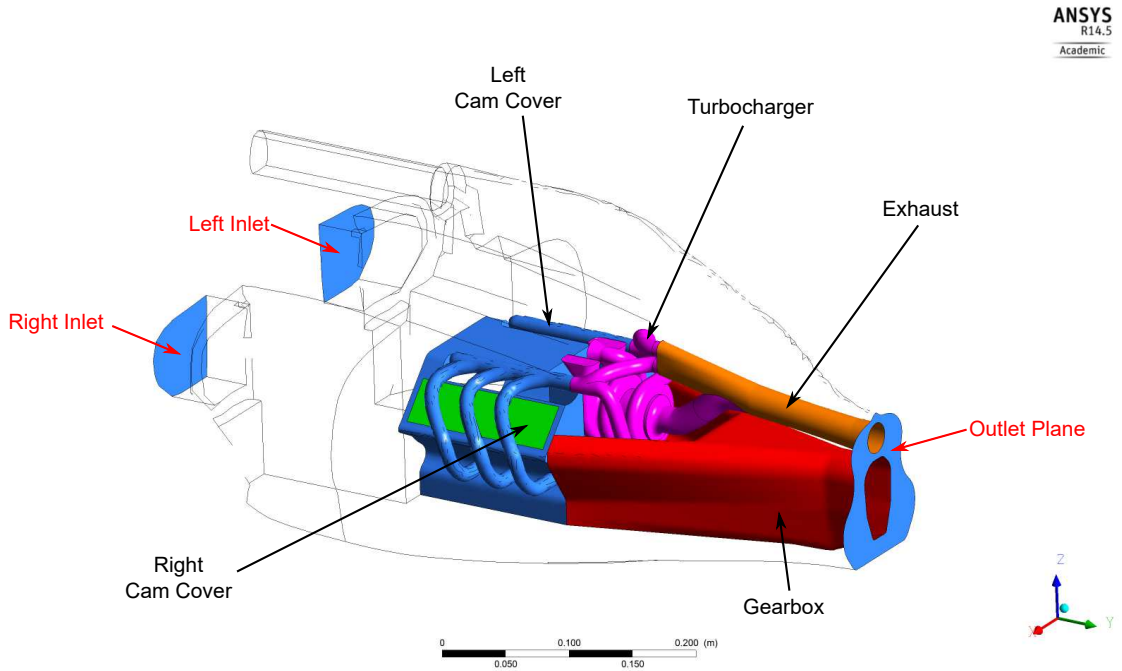


Figure 4.1: Overview of internal assembly and heat sources

4.1.2 Solver conditions and post processing

ANSYS Fluent was chosen to implement the numerical simulations, with a pressure based solver and the realizable $k-\epsilon$ turbulence model selected. Other turbulence models were investigated with minimal differences noticeable between the results. The use of prism layers (and an average y^+ value of under unity) on the boundary condition surfaces allowed the use of enhanced wall treatment (see Figure 4.3). The use of enhanced wall treatment blends the transition at the interface of the two-layer wall model (low Reynolds $k-\epsilon$ in the viscous sub-layer, and log-law otherwise). This

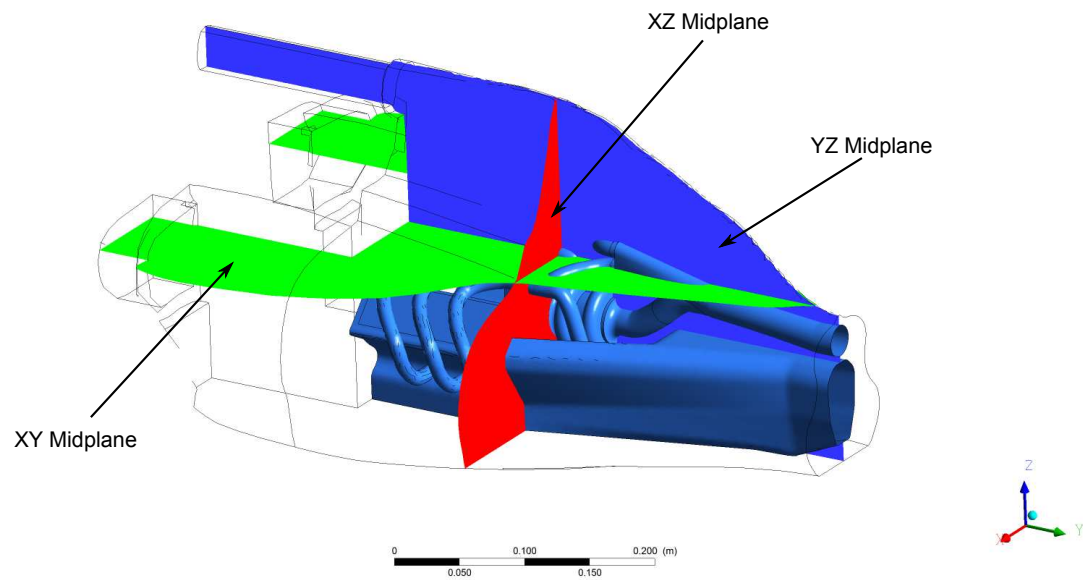


Figure 4.2: Planes used to process results of computational analyses

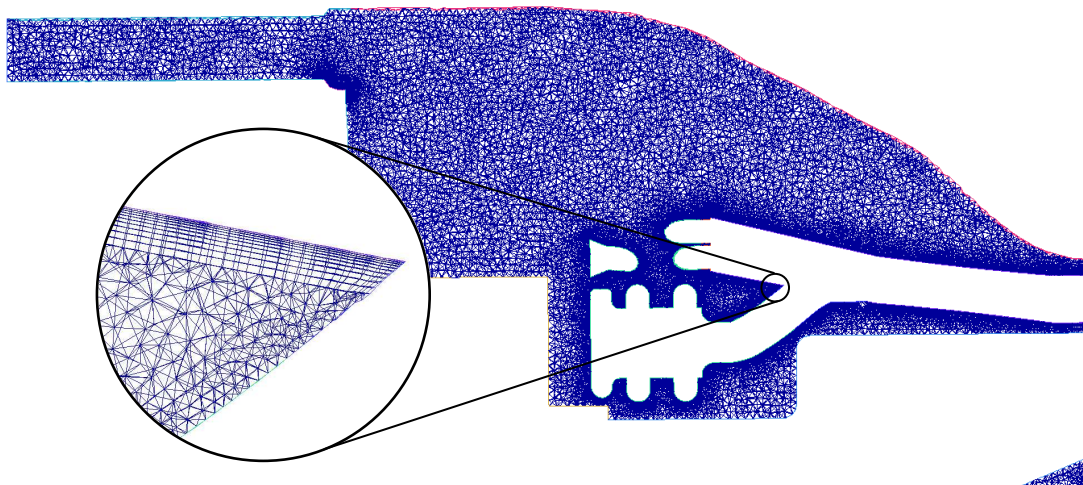


Figure 4.3: Detailed view of prism mesh on exhaust component in YZ mid plane (defined in Figure 4.2)

approach was chosen due to its wide use for internal cooling applications [75], [76], [77].

The results of the CFD analyses were then output to raw comma separated variable (CSV) files for post-processing in MATLAB. This method was chosen (as opposed to using a dedicated post-processor) in order to ensure parity in post-processing, and thus valid comparison, between the linear superposition method and direct CFD calculations. The results for both the linear superposition method and the CFD analyses were post-processed in MATLAB. Unprocessed nodal data was imported into a MATLAB graphical user interface (GUI) where the linear superposition method was applied.

4.2 Linear Scaling of Temperature Fields

4.2.1 Populating the temperature influence matrix

As described in Section 3.4, the temperature influence matrix is defined by running initial characterisation analyses. Five individual calculations were run with each heat source set to a heat flux boundary condition of 1 kW/m^2 in order to calculate the vector of temperature influence coefficients for each individual heat source (see Equation 3.7). These vectors were then combined to form the temperature influence matrix (as explained in Section 3.4). An inlet velocity of 2.5 m/s was chosen to match the experimental conditions. The temperature influence coefficients for some of the heat sources (shown in Figure 4.1) can be seen in Figures 4.4, 4.5 and 4.6.

One significant advantage of this approach is the ability to decouple the energy and momentum equations in the CFD computations, which allows the initial analyses to be completed quickly. Once the momentum equations have been solved to establish the velocity field within the domain, the solver effectively re-solves only the energy equations with the new thermal boundary conditions to establish the changes in the temperature field. Thus, when running the initial sets of calculations, only the first calculation (which includes the solving of both the momentum and energy equations) requires a full solution of the flow field. For the subsequent calculation of different thermal boundary conditions, given that the geometry and velocity field do not change

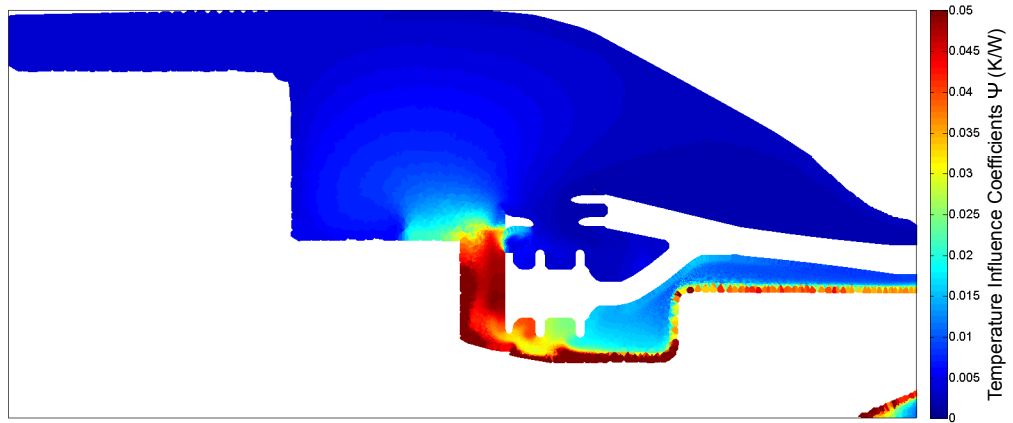


Figure 4.4: Temperature influence coefficients in the YZ mid plane generated from gearbox characterisation analysis

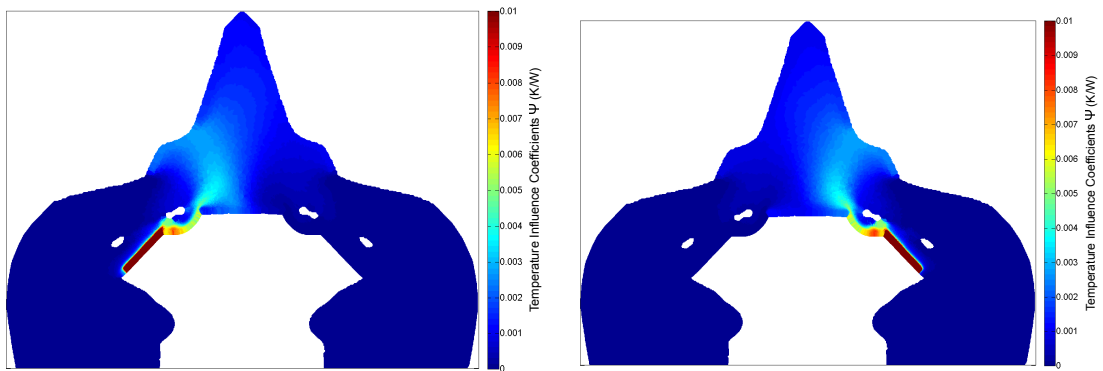


Figure 4.5: Temperature influence coefficients in the XZ mid plane generated from left and right engine cam cover characterisation analyses

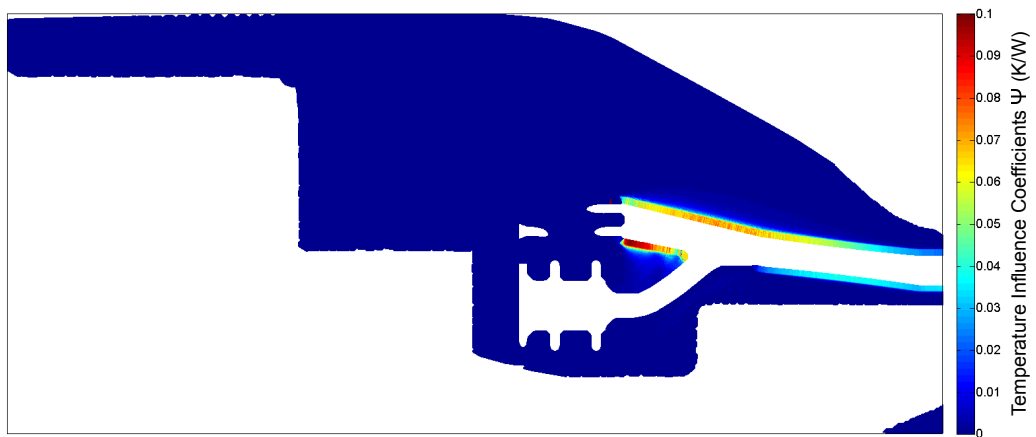


Figure 4.6: Temperature influence coefficients in the YZ mid plane generated from exhaust characterisation analysis

from the initial calculation, the time taken to solve just the energy equation (to a set residual accuracy) is a fraction of the time taken to complete the first calculation.

For the domain shown in Section 4.1.1, these subsequent calculations were completed within minutes as opposed to several hours for a full solution of the momentum and energy equations using 4 processing cores.

4.2.2 Superposition of individual temperature fields

Once the initial set of calculations are completed, it is then possible to use the equations described in Section 3.4 to predict the temperature field arising from a combination of thermal boundary conditions. In order to investigate this theory, the temperature influence matrix was defined using the results from a set of calculations with each heat source set to a heat flux boundary condition of 1 kW/m^2 (shown in Figures 4.4, 4.5 and 4.6). The author was then able to predict the temperature field that would arise in the domain if all the heat sources were enabled simultaneously by superposing the results from the initial calculations - the results of which can be seen in Figures 4.7, 4.8 and 4.9.

Every node in the computational mesh was represented in the temperature influence matrix, so that the 3-D temperature field could be predicted throughout the domain. Figures 4.7, 4.8 and 4.9 show the temperature fields at various planes throughout the domain for both the predicted temperature field using linear superposition and the explicit numerical analyses. It can be seen that the agreement between the predicted temperature fields using the linear superposition technique and the explicit numerical calculations is very good with the two solutions being almost indistinguishable.

The difference between the explicit numerical technique and the linear superposition method can be quantified by considering a comparison of the component surface temperatures. Accurate prediction of the component surface temperatures is particularly important if the output of the linear superposition model is subsequently used to calculate the effects of radiative heat transfer in the system or if the surface is used to dissipate heat into the engine bay - for example by cooling lubricant or thermally convected electronic components.

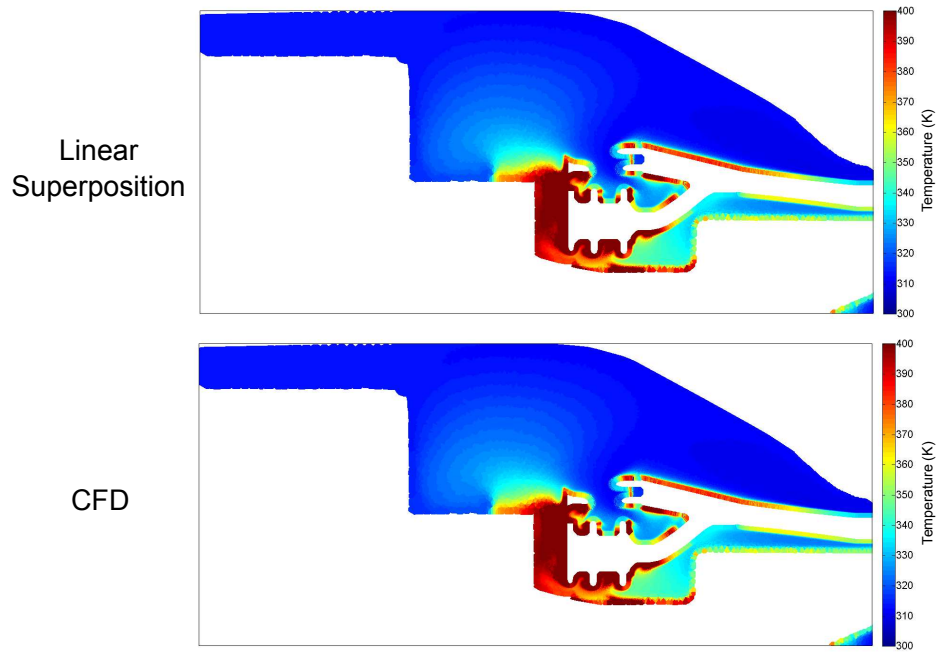


Figure 4.7: Comparison between superposition of individual analyses and explicit CFD at the YZ mid plane

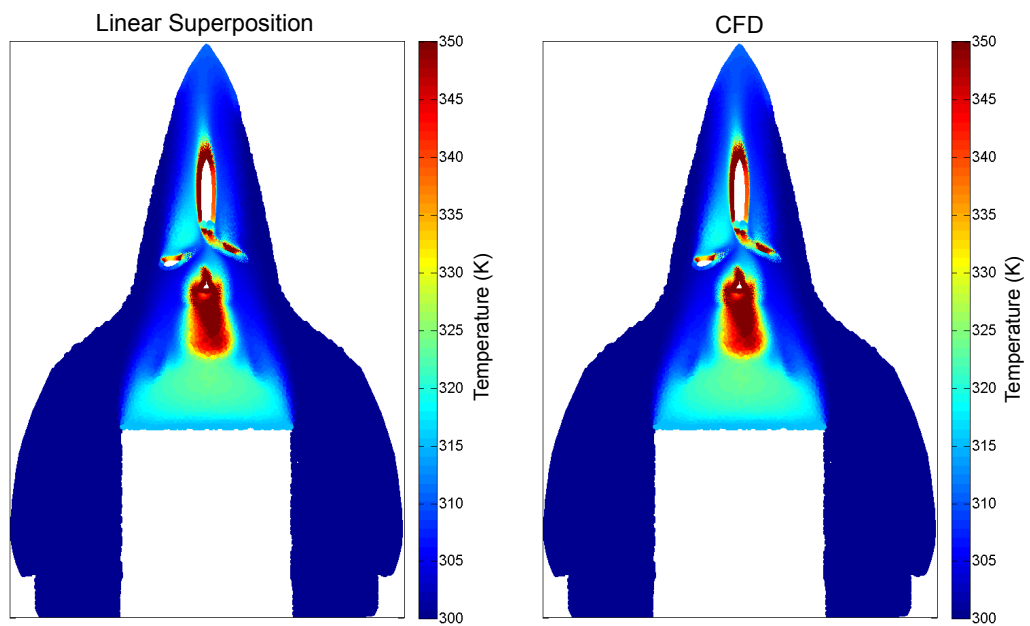


Figure 4.8: Comparison between superposition of individual analyses and explicit CFD at the XY mid plane

Table 4.1: Area averaged component surface ΔT for cases where a heat flux of 1 kW/m² is applied at the active heat source (where ΔT is defined as the the difference between component surface temperature and inlet air temperature)

Active Heat Source	Left Cam Cover Surface ΔT (K)	Right Cam Cover Surface ΔT (K)	Turbocharger Surface ΔT (K)	Exhaust Surface ΔT (K)	Gear-box Surface ΔT (K)
Left Cam Cover	42.809	0.037	0.725	0.630	0.419
Right Cam Cover	0.028	43.354	1.429	0.587	0.546
Turbocharger	0.194	0.235	95.248	9.080	3.627
Exhaust	0.002	0.002	0.595	46.473	0.152
Gearbox	0.111	0.135	14.194	3.006	57.924

Table 4.2: Comparison of area averaged component surface ΔT between summation of individual cases and simultaneous case (where ΔT is defined as the the difference between component surface temperature and inlet air temperature)

Case	Left Cam Cover Surface ΔT (K)	Right Cam Cover Surface ΔT (K)	Turbocharger Surface ΔT (K)	Exhaust Surface ΔT (K)	Gear-box Surface ΔT (K)
Summation of individual analyses	43.144	43.763	112.191	59.776	62.668
Simulation with all heat sources active	43.142	43.762	112.197	59.791	62.619
Absolute difference	0.002	0.001	0.006	0.005	0.049
<i>Percentage difference</i>	<i>0.005%</i>	<i>0.002%</i>	<i>0.005%</i>	<i>0.008%</i>	<i>0.078%</i>

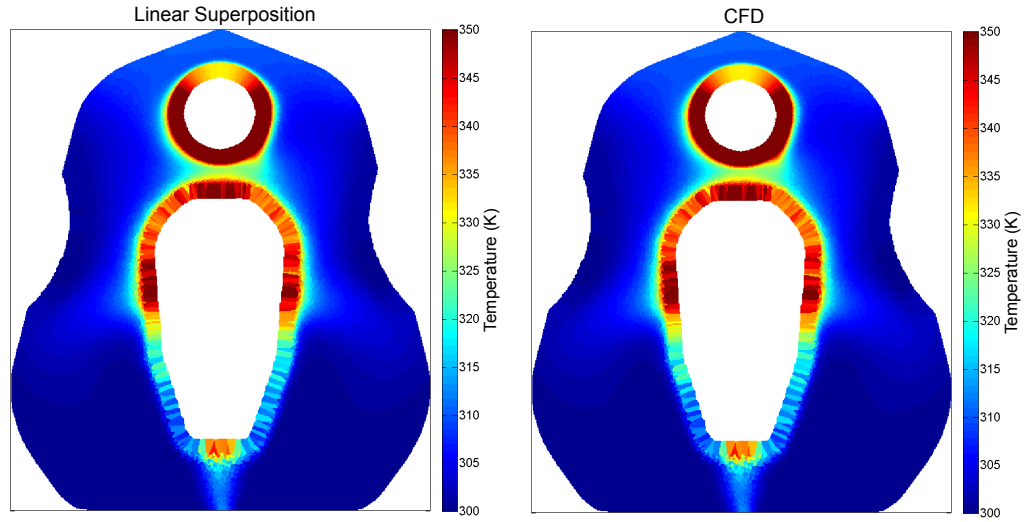


Figure 4.9: Comparison between superposition of individual analyses and explicit CFD at the outlet plane

Table 4.1 shows the average surface temperature of the heat sources in the system when a single heat source is set to a heat flux boundary condition of 1 kW/m^2 . As expected, the highest rise in surface temperature is seen on the surface of the active heat source (which can be seen on the top left to bottom right diagonal of Table 4.1). The magnitude of the surface temperature rise on each source for the case when it is not heated is indicative of the unheated heat source's geometric proximity to the active heat source and local direction of the coolant flow.

Table 4.2 shows a comparison between the summation of the component surface temperatures for the individual cases (i.e. the application of the linear superposition technique) and compares them to an analysis with all the heat sources active (which represent a numerical simulation with the same explicit boundary conditions as the summation of the individual cases). Table 4.2 shows that the results of both the linear superposition method and the explicit numerical simulation shows remarkably good agreement. The agreement between the two methods is within 0.1% for all cases.

4.2.3 Scaling of heat fluxes

It is also possible to predict the temperature fields that would arise if the heat flux from any one heat source is altered. As before, the same initial individual calculations were used as in Figures 4.4, 4.5 and 4.6 with each heat source individually set to a

heat boundary condition of 1 kW/m^2 . These were then combined, as in Section 4.2.2, to form the temperature influence matrix as described in Section 3.4. Using Equation 3.6, the author was then able to predict the temperature field with all the heat sources set to different heat flux boundary conditions simultaneously (as in Figures 4.7, 4.8 and 4.9). The result is a linear scaling of the superposed temperature fields seen in Figures 4.7, 4.8 and 4.9. As before, the results from this linear scaling approach were compared to computational analyses with the same explicit boundary conditions throughout the domain.

Figures 4.10 and 4.11 show a comparison between the temperature field predictions generated using the linear superposition technique and the computational analysis undertaken with the combined boundary conditions applied simultaneously. In this case, the temperature field has been predicted by superposing the five individual analyses with each heat source set to a 1 kW/m^2 heat flux boundary condition (as described previously), followed by a linear scaling operation. Linear superposition was then used to predict the resultant temperature field with all five heat sources set to a 2 kW/m^2 heat flux boundary condition simultaneously. As observed for the results in Section 4.2.3, it can be seen that the agreement between the two methods is very good, with very similar results throughout the domain.

This similarity in results between the linear superposition and the explicit computational analyses is repeated when the same method is used to generate a thermal field prediction for a 5 kW/m^2 set of heat flux boundary conditions as shown in Figures 4.12 and 4.13. Both quantitatively and qualitatively, the similarity in results shows that the linear superposition method is an accurate alternative to the more resource intensive computational analysis methods that are currently used. As before, the agreement between both methods was within 0.1% for all cases.

Application of the linear superposition method can be considered as the summation of individual temperature fields. Table 4.3 quantifies the discrepancies between the explicit CFD and the linear superposition method when superposing the individual component solutions.

Table 4.3 shows the mean temperature change (relative to the inlet temperature) throughout the domain where each heat source is set to a certain heat flux boundary

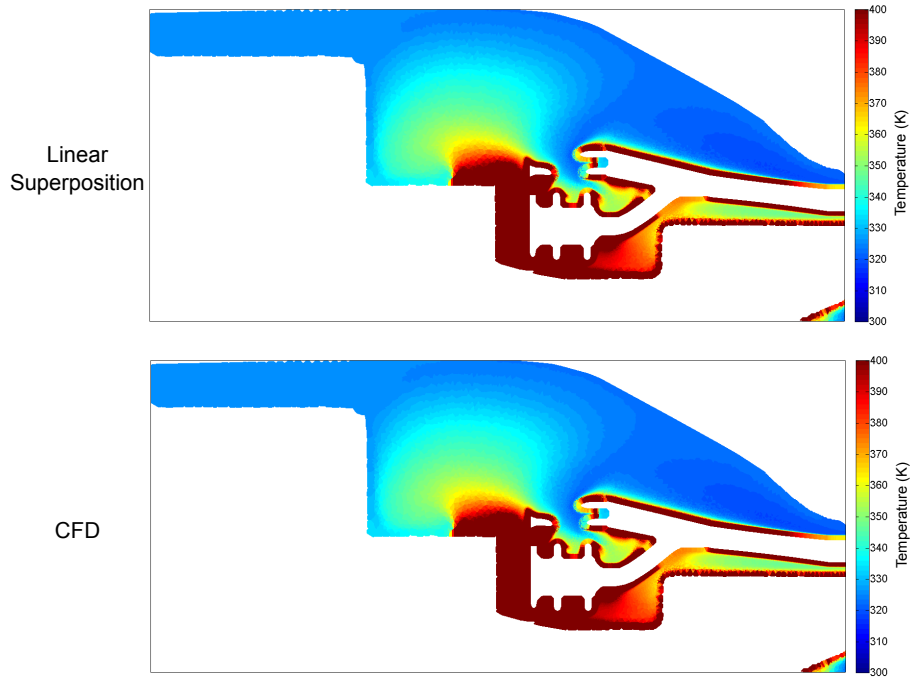


Figure 4.10: Comparison at YZ mid plane between the temperature field calculated using linear scaling from the 1 kW/m² solution to 2 kW/m² solution with the explicit 2 kW/m² CFD case

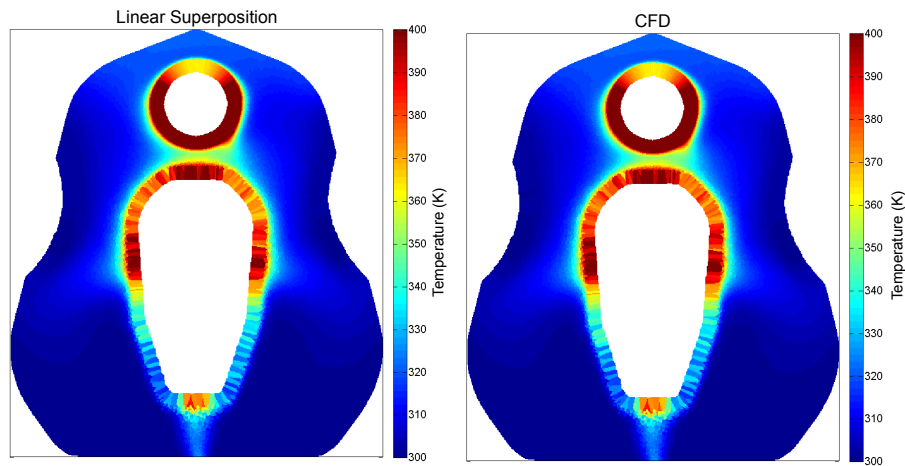


Figure 4.11: Comparison at the outlet plane between the temperature field calculated using linear scaling from the 1 kW/m² solution to 2 kW/m² solution with the explicit 2 kW/m² CFD case

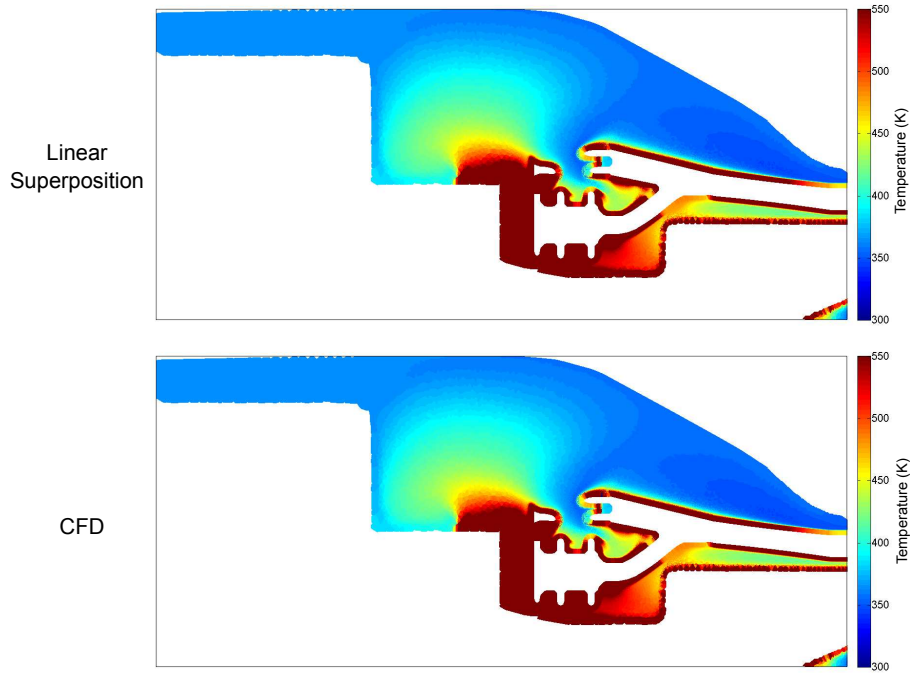


Figure 4.12: Comparison at YZ mid plane between the temperature field calculated using linear scaling from the 1 kW/m² solution to 5 kW/m² solution with the explicit 5 kW/m² CFD case

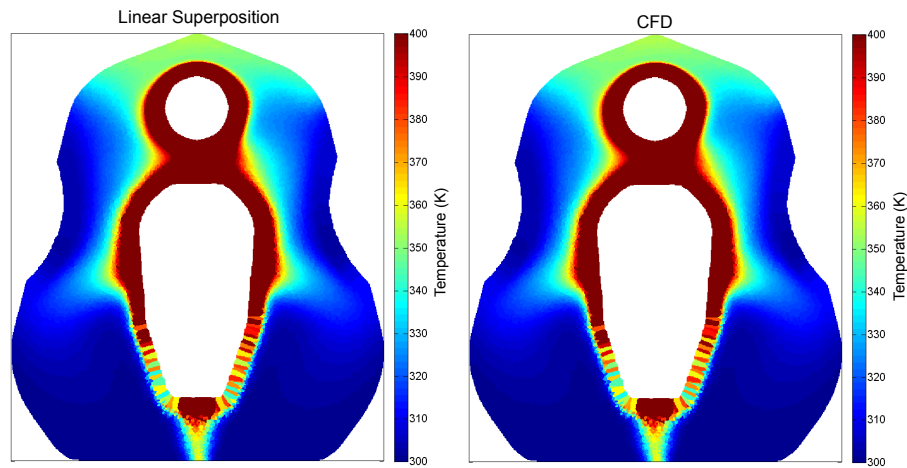


Figure 4.13: Comparison at the outlet plane between the temperature field calculated using linear scaling from the 1 kW/m² solution to 5 kW/m² solution with the explicit 5 kW/m² CFD case

Table 4.3: Individual and simultaneous heating comparison

Domain Average ΔT (K)	Heat Flux (W/m ²)			
	500	1000	2000	5000
Left Heater	0.202	0.405	0.810	2.027
Right Heater	0.197	0.392	0.783	1.957
Turbo Center	0.835	1.671	3.344	8.361
Turbine Inlet	0.487	0.972	1.941	4.855
Summation	<i>1.721</i>	<i>3.440</i>	<i>6.878</i>	<i>17.200</i>
Simultaneous Heating	<i>1.720</i>	<i>3.441</i>	<i>6.882</i>	<i>17.206</i>
Percentage Difference	0.058	0.029	0.058	0.035

condition. These individual contributions are then summed together and compared to a computational simulation where the thermal boundary conditions of the heat sources are set to the same value simultaneously. This is then repeated with the heat source boundary conditions set to a variety of different heat flux values. It can be seen that the percentage difference between the linear superposition method and the computational simulation with the same explicit boundary conditions is less than 0.1% in all cases.

4.3 The Effect of Changing Inlet Velocity

As outlined in Section 3.5, the linear superposition technique sets out an approach to take into account changes in the inlet velocity. As in Section 4.2.1, an initial set of analyses (with each heat source set to a heat flux boundary condition of 1 kW/m² independently) was used to form the base of the thermal model. The inlet velocity for each of these initial analyses was set to 9 m/s. For a set of three thermal boundary conditions, the temperature field was then scaled using Equation 3.11 in order to predict the temperature field that would occur at lower inlet velocities.

This initial set of analyses is first used to calculate the non-dimensional temperature influence coefficients as defined in Equation 3.10. The inlet velocity for these calculations is no longer arbitrary but is defined as the datum velocity for any predictions

made from these initial set of analyses. Three sets of thermal boundary conditions were used in order to compare the linear superposition and CFD methods. These were as follows:

1. All components active
2. Exhaust part active
3. Turbocharger active

The aforementioned set of boundary conditions were considered good examples of different types of heat addition into the domain. The same method of comparison as before is used with the scaling using linear superposition compared against CFD analyses with the same explicit boundary conditions. In order to further analyse the agreement between the two methods, the domain is separated into discrete bins in the YZ mid plane. The mean temperature change in each bin is then investigated and plotted to give the temperature distribution along the length of the YZ midplane. The average difference across all the bins and maximum observed difference over all the bins between the linear superposition and CFD methods is then calculated.

4.3.1 Linear mass flow scaling

The first set of calculations were run with an inlet velocity of 9 m/s with linear mass flow scaling (as specified in Equation 3.11) applied in order to predict the temperature field that would result from lower inlet velocities using the three sets of thermal boundary conditions described in Section 4.3.

Figures 4.14, 4.15, and 4.16 show the predicted temperature field that results from the set of three thermal boundary conditions (described in Section 4.3) at different inlet velocities (scaling down from the datum inlet velocity of 9 m/s). It can be seen that the agreement between the two methods of predicting the temperature field reduces as the scaling factor increases and the inlet velocity diverges from the datum case (9 m/s).

The majority of the above difference is likely to be caused by changes in the velocity flow field (as a result of the decrease in inlet velocity) which are not captured in the

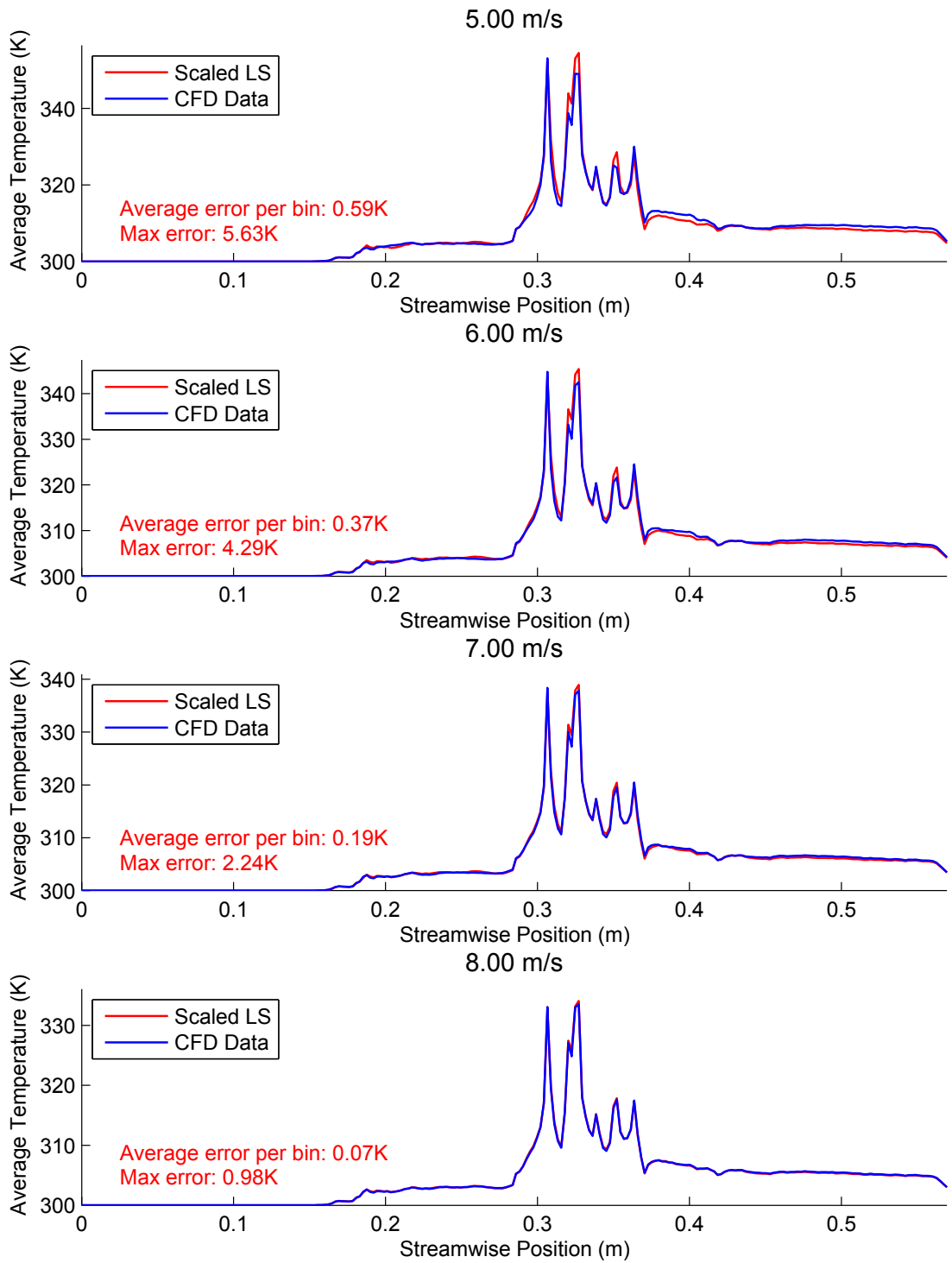


Figure 4.14: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method and explicit CFD for the all heat sources active at different inlet velocities (scaled from the 9 m/s datum velocity)

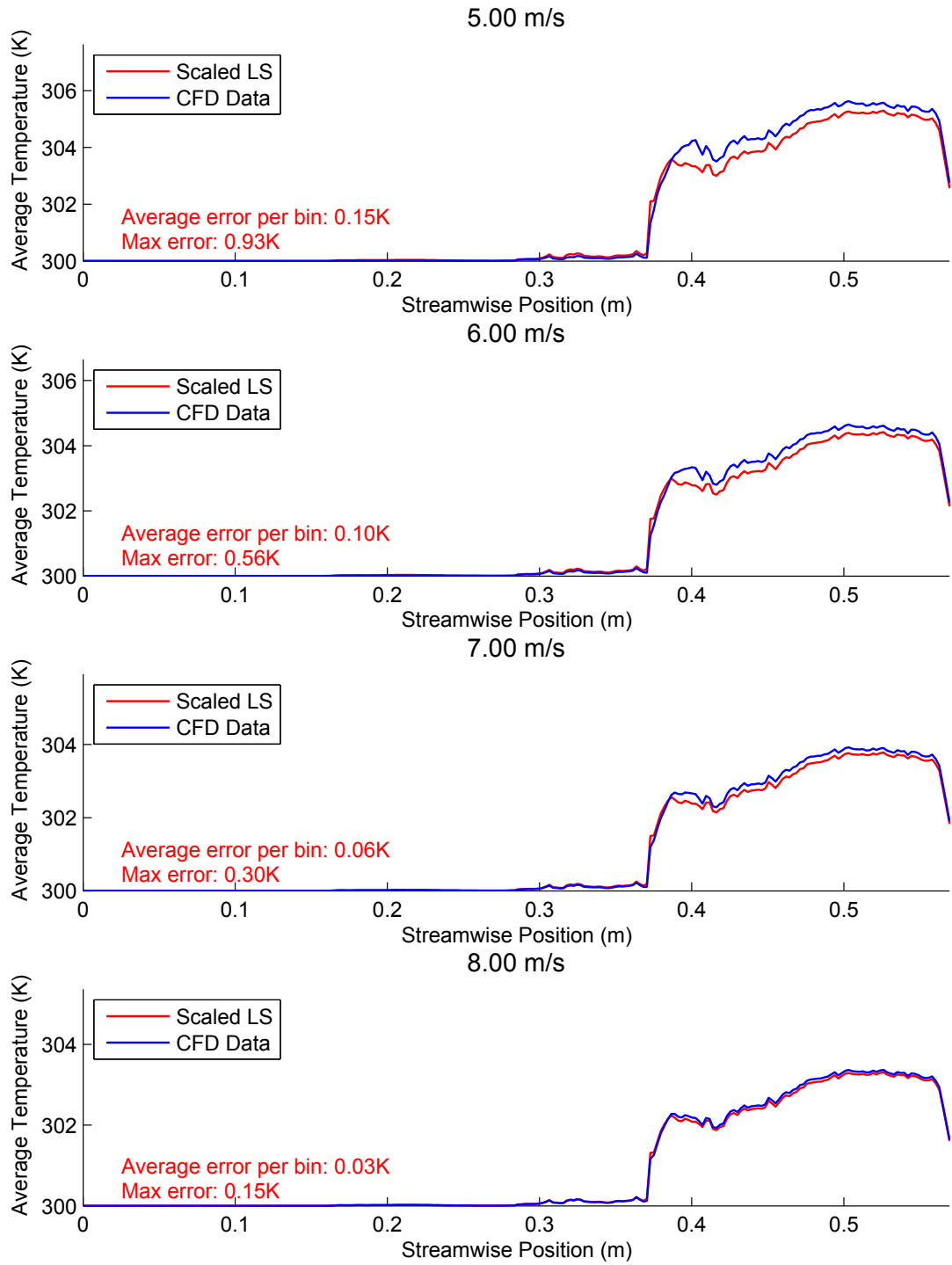


Figure 4.15: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method and explicit CFD for the exhaust heat source active at different inlet velocities (scaled from the 9 m/s datum velocity)

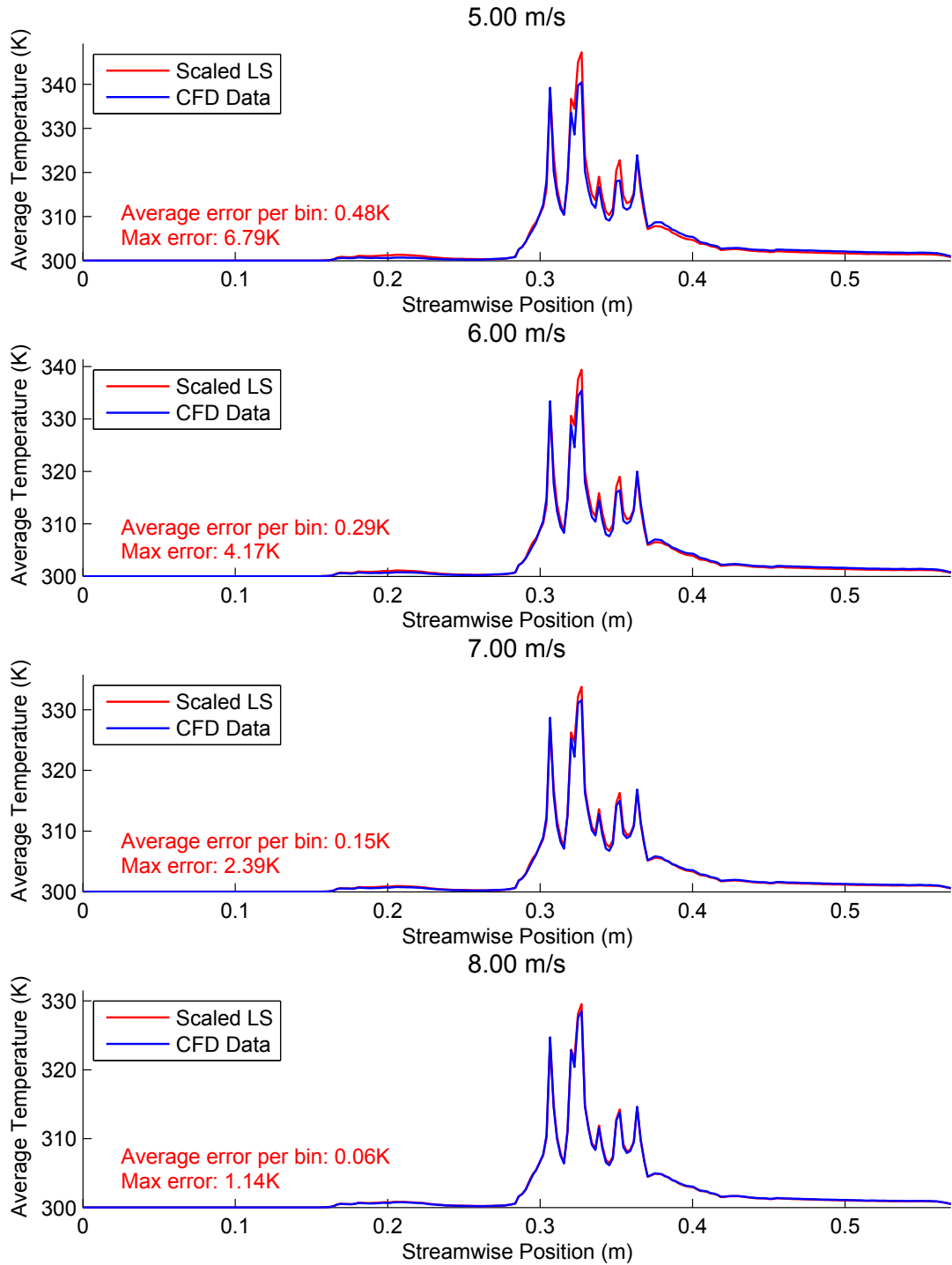


Figure 4.16: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method and explicit CFD for the turbocharger heat source active at different inlet velocities (scaled from the 9 m/s datum velocity)

linear superposition method. Another possible explanation is that the assumption of negligible Reynolds number dependency in Equation 3.11 is invalid (see Section 3.5).

In addition to this, as there is large variation in the velocity field throughout the system (owing to the geometrical complexity and large variations in cross sectional area throughout the domain), it is possible that in low velocity areas, the Grashof-Reynolds number ratio increases sufficiently such that the heat transfer in the area can no longer be considered as forced convection dominant, thus invalidating use of the linear superposition method in certain areas.

4.3.2 Scaling using Stanton number dependency

It was shown in Section 4.3.1 that the discrepancies between the results of the linear superposition method and the explicit numerical simulations increase as the inlet mass flow and datum case mass flow diverge. These discrepancies are partially due to the assumption (in Section 3.5) that the dependency of non-dimensional temperature change on Reynolds number can be considered negligible.

Section 3.6 hypothesised a correction factor to account for the Reynolds number dependency (see Section 3.3) by drawing an analogy to the simple case of a ducted heat exchanger with an isothermal wall (see Equation 3.20).

Having investigated the linear mass flow scaling as described in Equation 3.11, the same set of initial calculations (9 m/s inlet velocity and 1 kW/m² heat flux emissions, described in Section 4.2.1) were used to investigate whether the discrepancies seen in Figures 4.14, 4.15, and 4.16 could be reduced by taking into account the effect of Reynolds number dependency on dimensionless temperature using the procedure described in Section 3.3.

In order to apply the theory described in Equation 3.20, the non-linear correction factor was applied to the temperature fields that had already been linearly scaled to match the thermal and kinematic inlet boundary conditions (Figures 4.14, 4.15, and 4.16) where the temperature fields were predicted for reduced inlet flow speeds from the initial speed of 9 m/s.

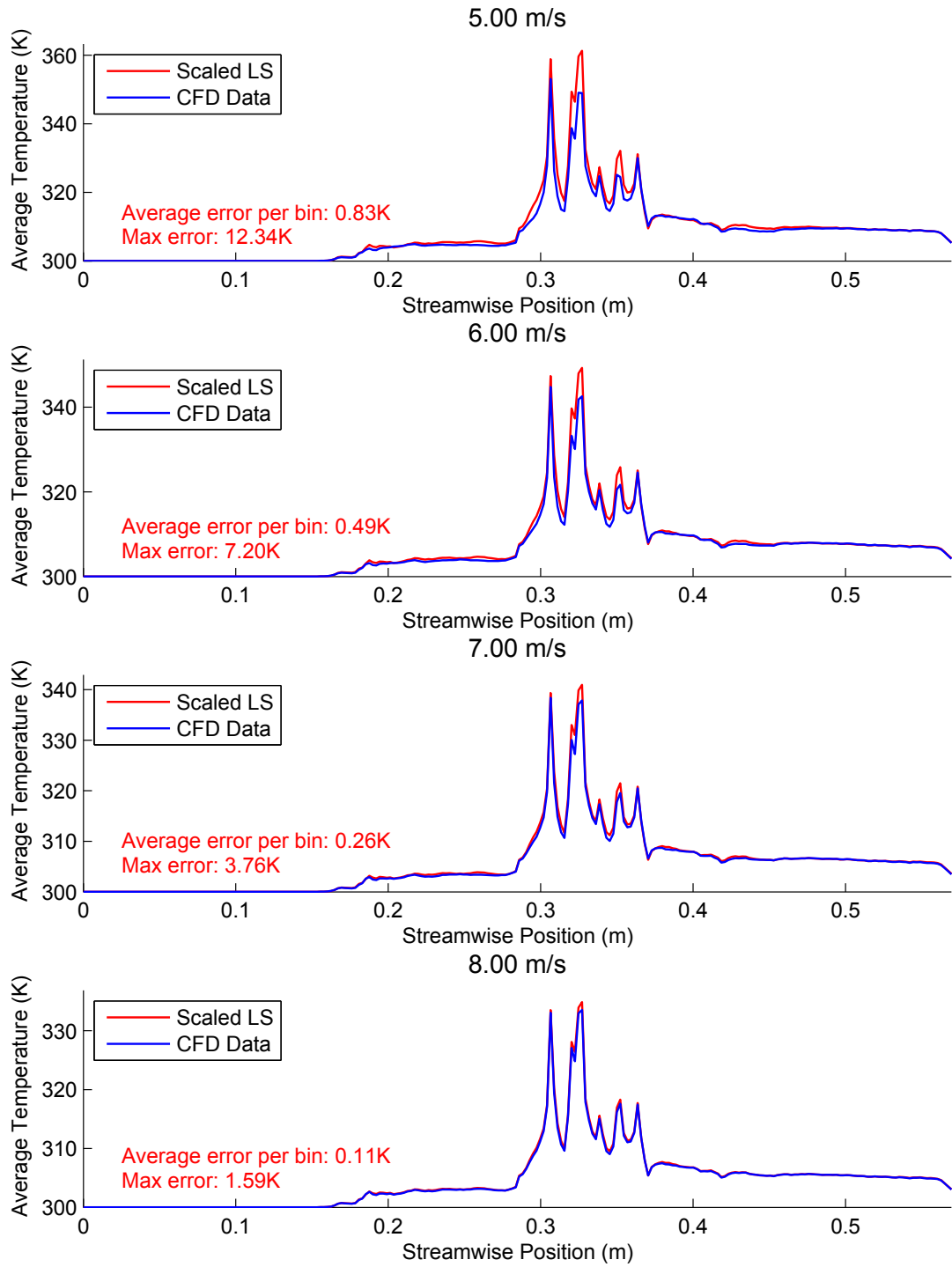


Figure 4.17: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method (with Stanton number correction applied, see Equation 3.17) and explicit CFD for the all heat sources active at different inlet velocities (scaled from the 9 m/s datum velocity)

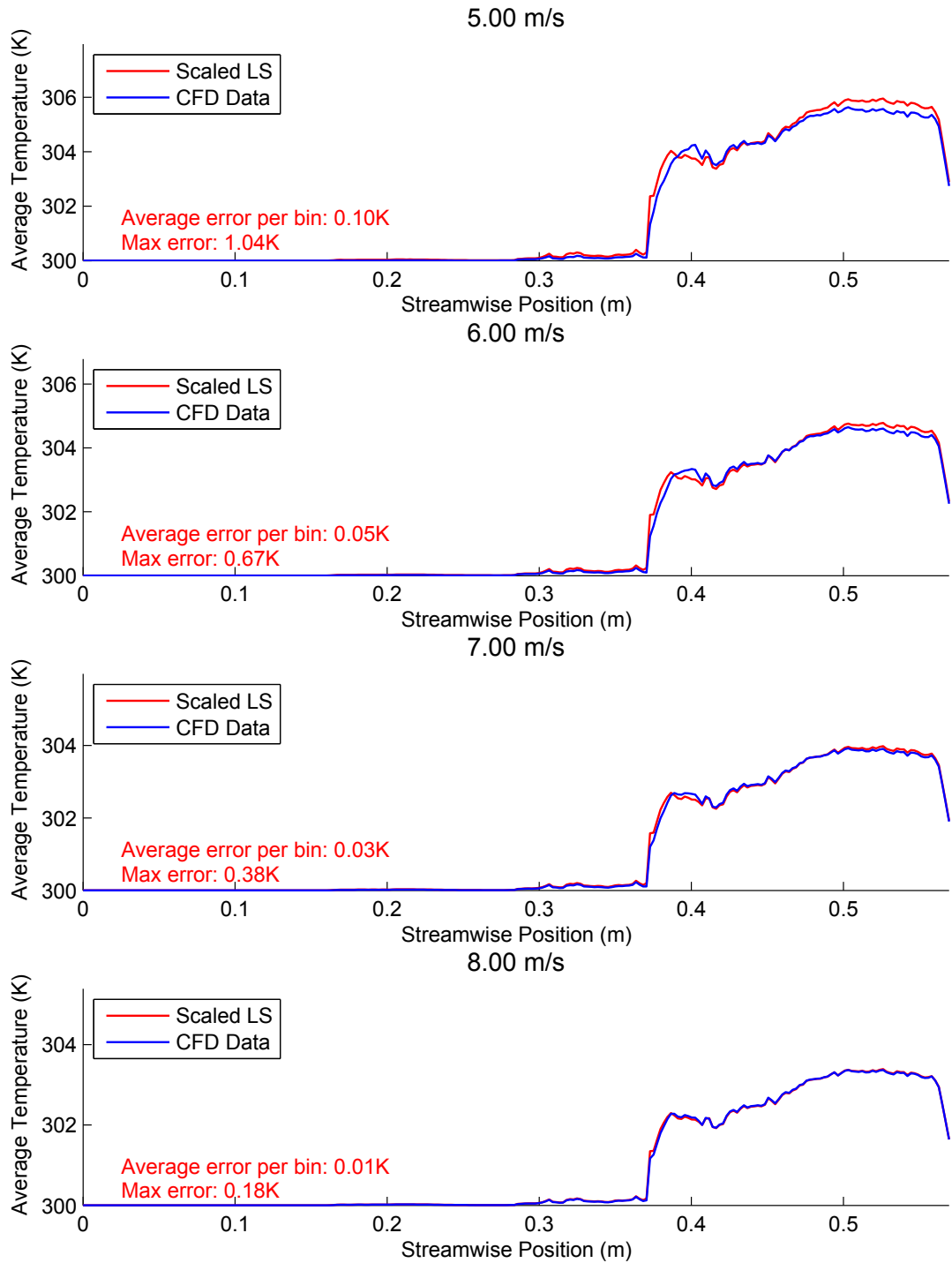


Figure 4.18: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method (with Stanton number correction applied, see Equation 3.17) and explicit CFD for the exhaust heat source active at different inlet velocities (scaled from 9 m/s the datum velocity)

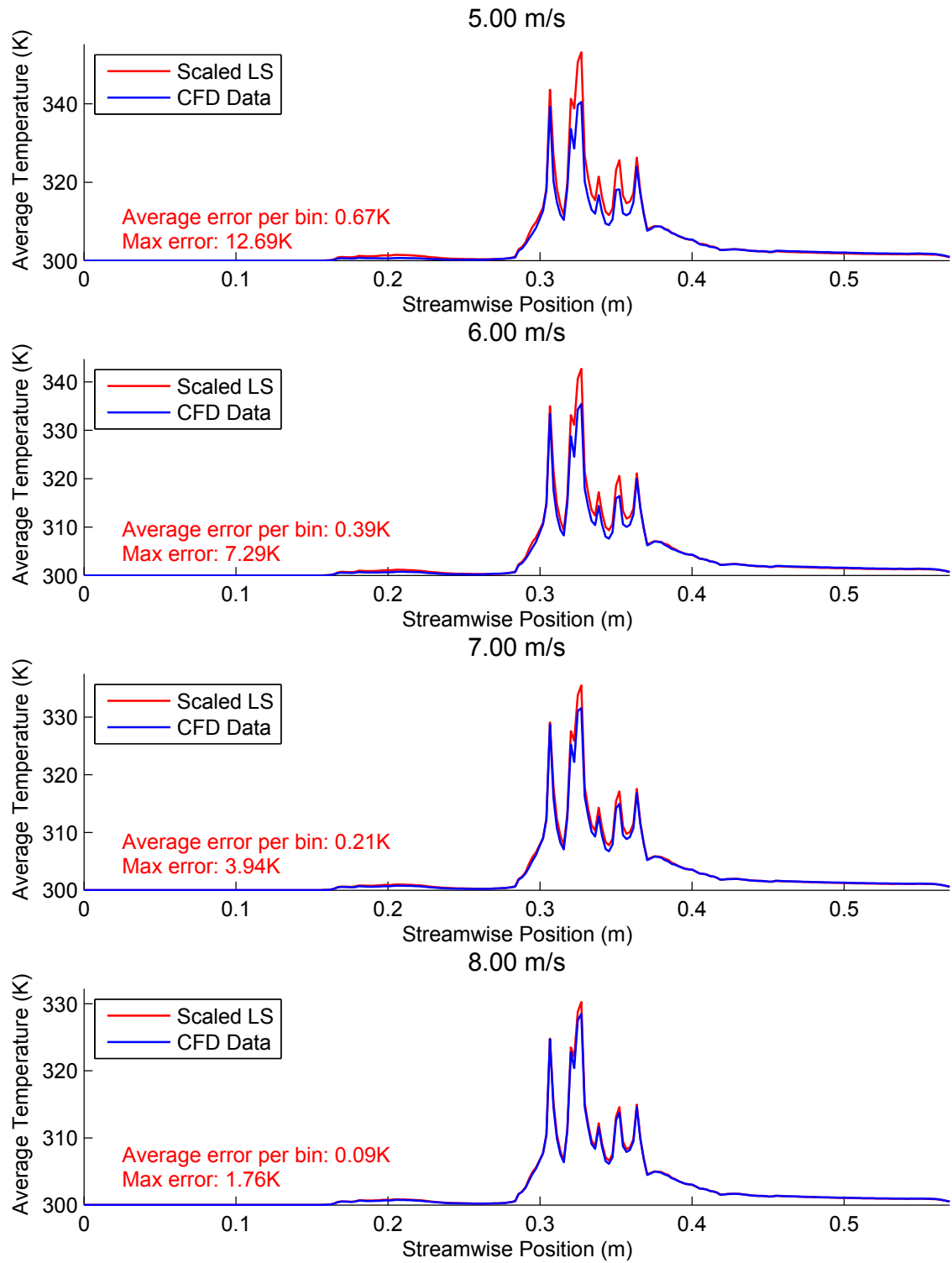


Figure 4.19: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method (with Stanton number correction applied, see Equation 3.17) and explicit CFD for the turbocharger heat source active at different inlet velocities (scaled from the 9 m/s datum velocity)

Figures 4.17, 4.18, and 4.19 show the results of applying the non-linear scaling (described in Equation 3.20) to the temperature fields described in Figures 4.14, 4.15, and 4.16. From a comparison of the two temperature fields in Sections 4.3.1 and 4.3.2, it is possible to identify the effect of applying the non-linear correction along the streamwise direction. It can be seen that the non-linear correction exacerbates the error observed after the linear mass flow scaling has been applied, with the most significant discrepancies at peak temperatures and in regions of large temperature change (i.e. the streamwise location of any heat sources in the domain) being increased after the application of the non-linear correction. An example of this can be seen at the 0.33 m streamwise position in the active turbocharger heat source case (see Figure 4.19).

However, it is equally evident that in areas of small temperature change, where the temperature profile is steady (i.e. there is no heat flux addition, either downstream or upstream of the active heat sources), the non-linear Stanton number scaling had the effect of reducing the discrepancy between the linear superposed predicted thermal field and the explicit numerical analyses.

4.3.3 Temperature gradient weighted Stanton number scaling

It can be seen from Figures 4.17, 4.18, and 4.19 that the non-linear correction (described in Section 3.6) increases the error observed in areas of high temperature increase, whilst reducing the error in areas where the temperature profile exhibits little change over the streamwise direction. In this section, the author investigates the benefit of applying a varying weighting factor to the application of the non-linear correction in order to correct for this variation. The approach is based on the empirical results shown in Section 4.3.2.

$$T_i = T_{in} + \Delta T_i \quad (4.1)$$

$$T_i = T_{in} + \Delta T_i \left(\frac{u_{in}}{u_{datum}} \right)^{-0.2} \quad (4.2)$$

$$T_i = T_{in} + \Delta T_i (1 + w_i [\left(\frac{u_{in}}{u_{datum}} \right)^{-0.2} - 1]) \quad (4.3)$$

Equation 4.1 describes the linear mass flow scaling as used in Section 4.3.1. The non-linear correction is then applied using Equation 4.2 to produce the results shown in Section 4.3.2. Equation 4.3 shows the new variable weighting, w_i , applied to the non-linear scaling factor in order to modulate the effect of the Reynolds number correction.

From the discussion above, it is evident from the results shown in Section 4.3.2 that the non-linear correction should only be applied in areas of low streamwise temperature change, and simple linear scaling (see Equation 4.1) should be used elsewhere. These areas of low streamwise temperature change can be considered to correspond to areas away from the active heat sources in the domain. As the shape of the temperature profile does not change between applications of the linear superposition method (geometry and thermal boundary conditions do not vary at this stage), it is possible to use features of the streamwise temperature profile in order to inform the weighting function.

The gradient of the streamwise temperature profile is the parameter chosen by the author to determine the weighting of the non-linear correction at any point in the domain. This approach results in Equation 4.4, where the weighting has been normalised using the maximum gradient in the streamwise temperature profile (where x is the streamwise coordinate) in order to bound the effect of the non-linear correction.

$$w_i = 1 - \left[\frac{\left(\frac{dT}{dx}\right)_i}{\left(\frac{dT}{dx}\right)_{max}} \right] \quad (4.4)$$

Figures 4.20, 4.21, and 4.22 show the effect of applying the weighted non-linear correction to the results seen in Figures 4.17, 4.18, and 4.19. It can be seen that both the peak and general level of discrepancies between the predicted temperature fields using linear superposition and the explicit numerical analyses have been reduced by modulating the weight of non-linear Stanton number correction using Equation 4.3. As before, the discrepancy between the two methods of analysis increases as the inlet velocity diverges from the datum case which is to be expected.

The reduction of error in the areas of low temperature change in the streamwise profile indicates that the application of the non-linear correction (Equation 4.2) is

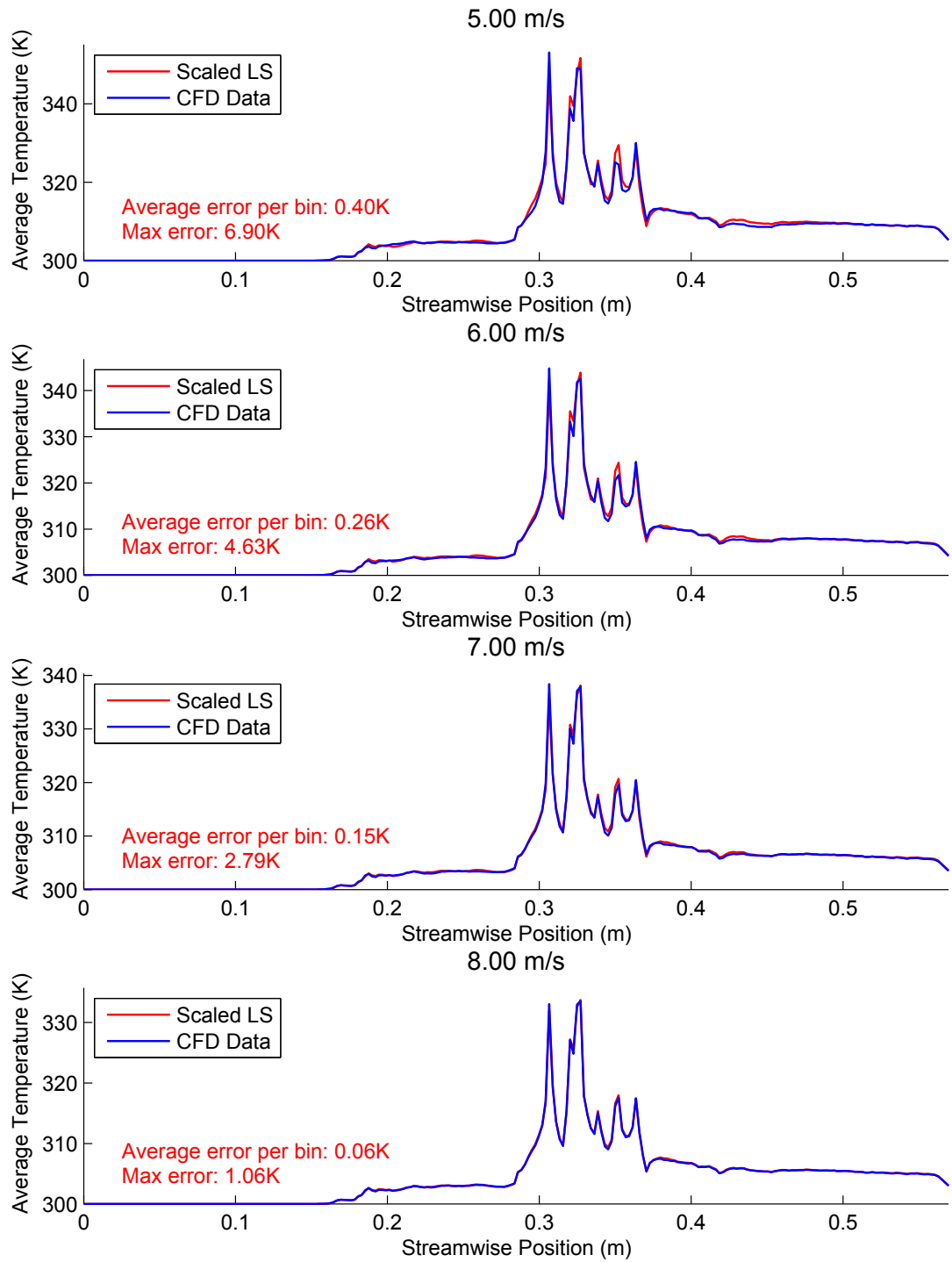


Figure 4.20: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method (with weighted Stanton number correction applied) and explicit CFD for the all heat sources active at different inlet velocities (scaled down from 9 m/s datum velocity)

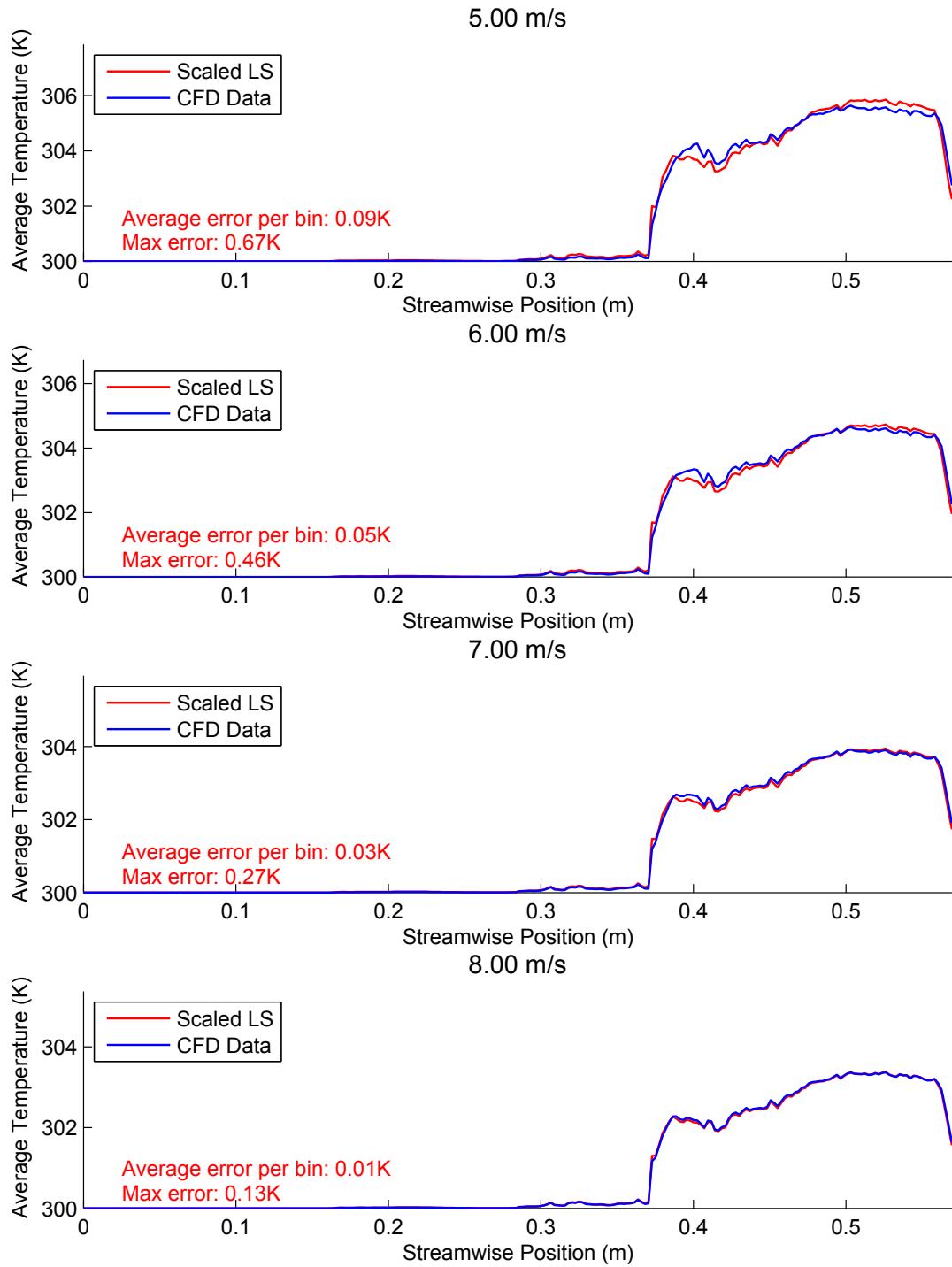


Figure 4.21: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method (with weighted Stanton number correction applied) and explicit CFD for the exhaust heat sources active at different inlet velocities (scaled down from 9 m/s datum velocity)

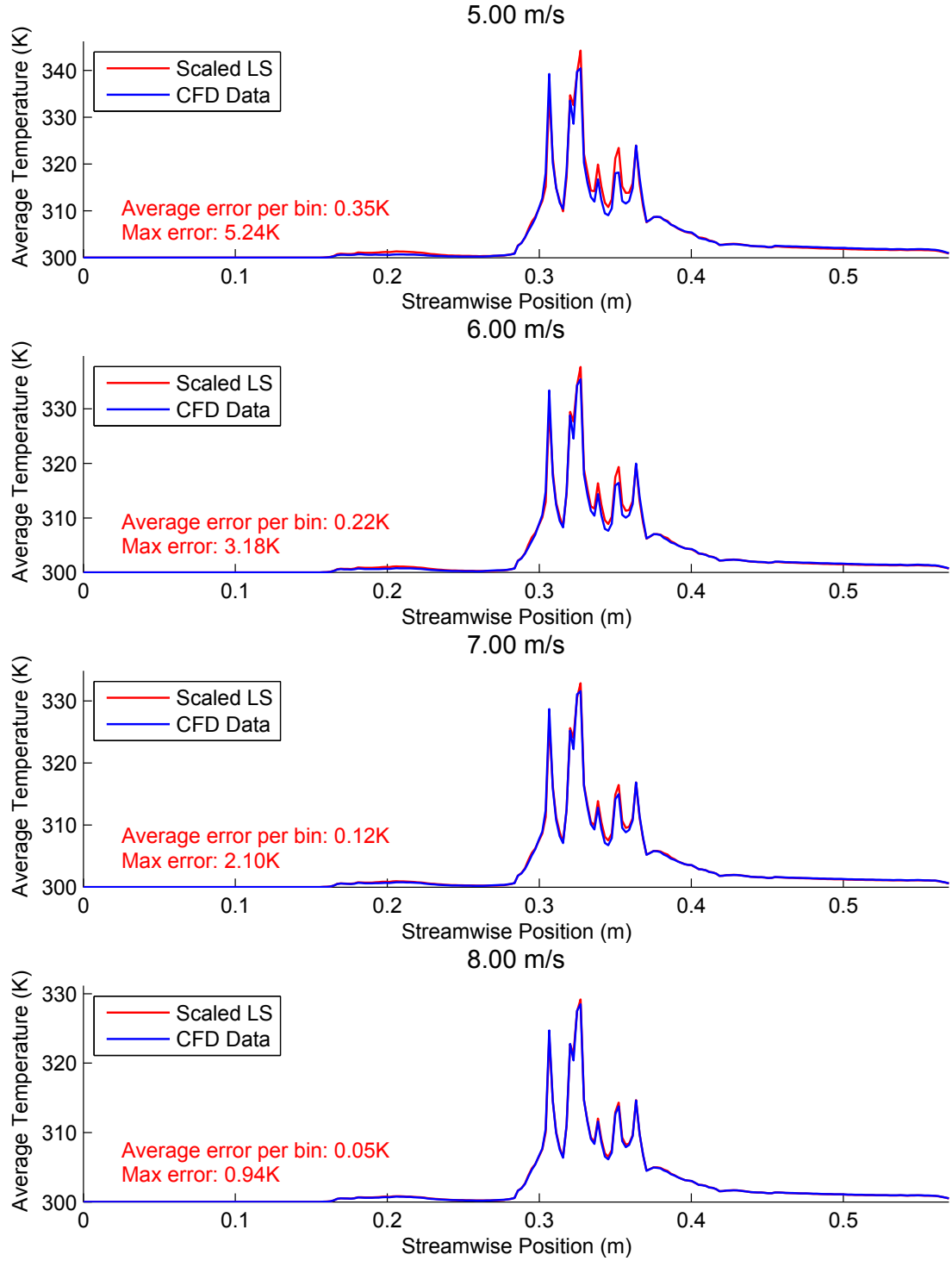


Figure 4.22: Comparison of the temperature profile, averaged across sections 11 mm thick, perpendicular to the Y axis, between the linear superposition method (with weighted Stanton number correction applied) and explicit CFD for the turbocharger heat sources active at different inlet velocities (scaled down from 9 m/s datum velocity)

valid when the heat addition from the active surfaces is mixed into the gas flow.

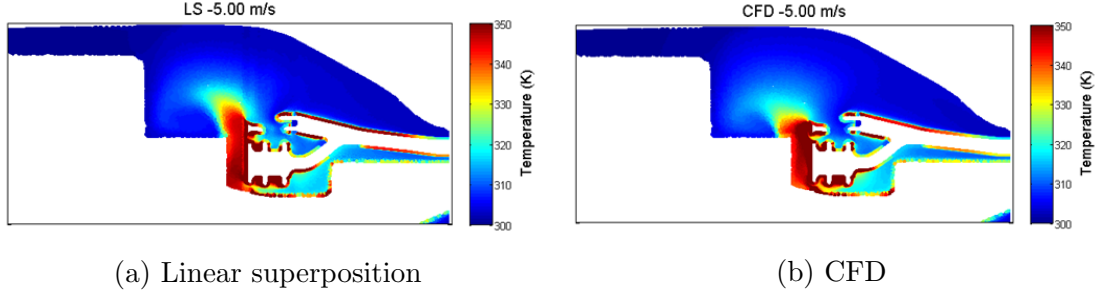


Figure 4.23: Comparison between weighted linear superposition with non-linear Stanton number correction and explicit numerical analysis at 5 m/s inlet velocity from a datum 9 m/s inlet velocity

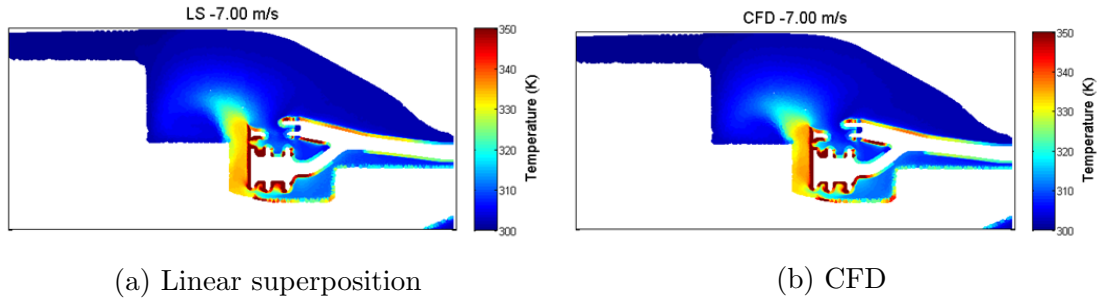


Figure 4.24: Comparison between weighted linear superposition with non-linear Stanton number correction and explicit numerical analysis at 7 m/s inlet velocity from a datum 9 m/s inlet velocity

Figures 4.23 and 4.24 show the difference in temperature field for the 5 m/s and 7 m/s inlet velocity cases, which have been scaled from a 9 m/s inlet velocity datum case using the weighted non-linear correction described by Equation 4.3. It can be seen that the magnitude of the temperature field is well captured by the non-linear scaling, but there are clear discrepancies between the locations of certain temperature features in the domain which is indicative of changes in the velocity flow field.

Table 4.4 shows the quantitative differences between using the linear mass flow scaling system, the linear mass flow scaling system with the non-linear correction and the linear mass flow scaling system with a weighted non-linear correction. Table 4.4 shows both the average and maximum temperature differences between the application of the mass flow scaling system and the explicit numerical simulations for the three

methods.

As observed in Section 4.3.2, the non weighted application of the non-linear correction tends to increase the maximum error observed in regions of heat application. As mentioned earlier, this is due to the non-linear correction exacerbating error in areas of high temperature change in the streamwise profile which naturally coincides with the streamwise location of a surface with an active thermal boundary condition. From Table 4.4, it can be seen that for the majority of cases, the mass flow scaling with the weighted non-linear correction gives the best balance of reduction in both maximum and average errors.

Table 4.4: Comparison of mass flow scaling methods (using a datum case of 9 m/s inlet speed)

	Inlet Speed (m/s)	Linear mass flow scaling		Mass flow scaling with non-linear correction		Mass flow scaling with weighted non-linear correction	
		<i>Average</i> ΔT (K)	<i>Max</i> ΔT (K)	<i>Average</i> ΔT (K)	<i>Max</i> ΔT (K)	<i>Average</i> ΔT (K)	<i>Max</i> ΔT (K)
All Active	5	0.59	5.63	0.83	12.34	0.40	6.90
	6	0.37	4.29	0.49	7.20	0.26	4.63
	7	0.19	2.24	0.26	3.76	0.15	2.79
	8	0.07	0.98	0.11	1.59	0.06	1.06
Exhaust	5	0.15	0.93	0.1	1.04	0.09	0.67
	6	0.10	0.56	0.05	0.67	0.05	0.46
	7	0.06	0.30	0.03	0.38	0.03	0.27
	8	0.03	0.15	0.01	0.18	0.01	0.13
Turbocharger	5	0.48	6.79	0.67	12.69	0.35	5.24
	6	0.29	4.17	0.39	7.29	0.22	3.18
	7	0.15	2.39	0.21	3.94	0.12	2.10
	8	0.06	1.14	0.09	1.76	0.05	0.94

4.4 Investigation of Velocity Scaling on Alternative Geometry

In an effort to provide a more fundamental analysis of the velocity scaling technique, an additional computational model was created. The geometry consisted of a converging section with two sets of bars included in the domain aligned in lateral and vertical orientations (shown in Figure 4.25). A 1 kW/m^2 heat flux boundary condition was specified at the surface of each set of bars.

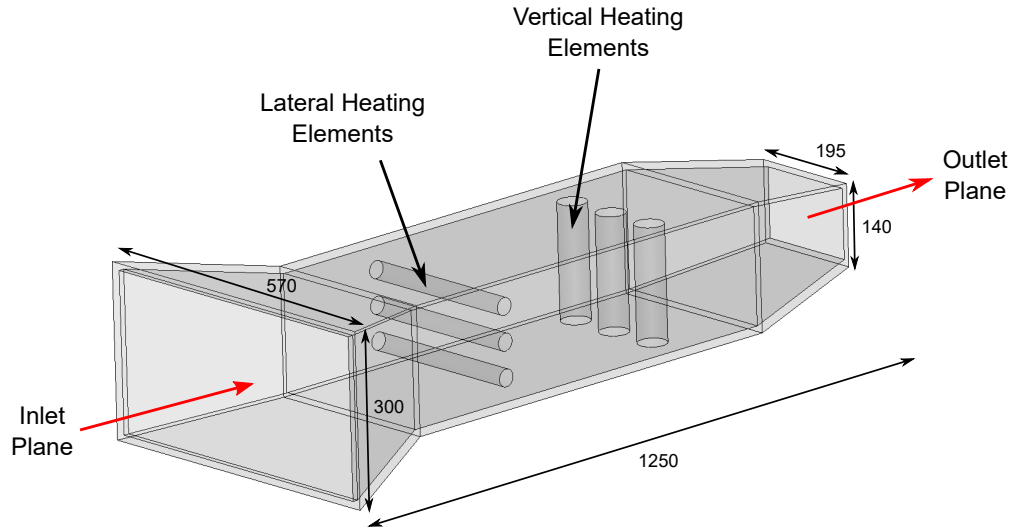


Figure 4.25: Overview of converging section computational model

As before, the mesh consisted of both tetrahedral and prism elements and comprised approximately 2.6 million cells. Prism cells were used on the thermal boundary conditions surfaces in order to provide a y^+ value under unity on these surfaces. The realisable $k-\epsilon$ turbulence model with enhanced wall treatment was used. The datum calculations were run at 9 m/s , with the different velocity scaling methods used to predict the temperature field at reduced velocities.

Figure 4.26 shows the error observed between the linear mass flow scaling method and CFD analyses with the same explicit boundary conditions for varying inlet velocities (scaled from the 9 m/s datum velocity). It can be seen that the maximum error is observed at the same streamwise locations as the heated lateral and vertical bars. This

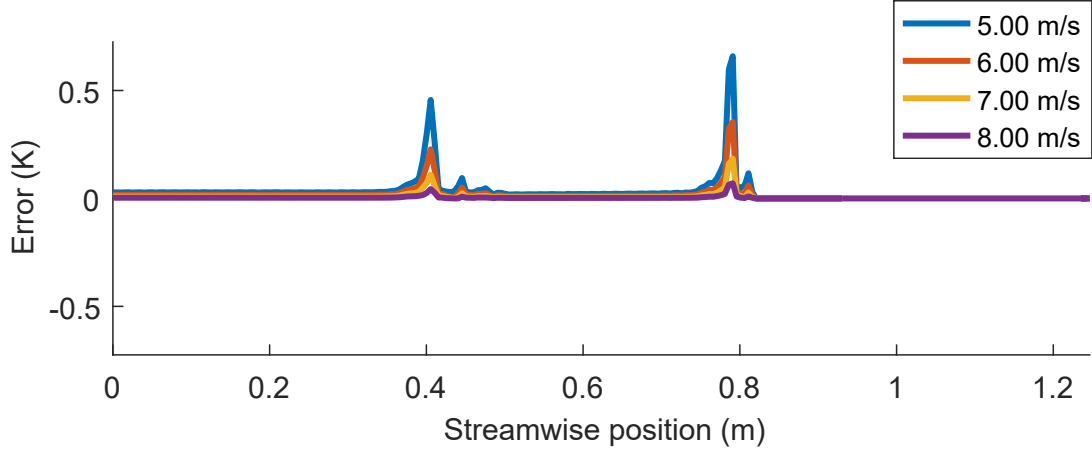


Figure 4.26: Error between linear mass flow scaling method and CFD analyses at varying inlet velocities (scaled from a 9 m/s datum velocity)

phenomenon of peak error occurring where heat is supplied mirrors the underhood race car geometry, where the location of peak error correlated with the location of heat addition within the domain. Figure 4.26 shows that the profile of the error is similar between the different inlet velocities with the magnitude proportional to the difference between the inlet velocity and datum velocity. However, it should be noted that the magnitude of the error is not linearly proportional to this difference as the error appears to increase with the square of the difference.

Figure 4.27 shows the effect of both the non-linear correction and weighted non-linear correction for a 5 m/s inlet velocity (scaled down from the 9 m/s datum velocity). The peak error can be observed at the positions where the peaks in the temperature profile occur, which correlate with the streamwise position of the heated lateral and vertical bars, where the linear superposition prediction can be seen to over predict the temperature. At these locations, it can be seen that the application of the non-linear correction then causes the linear superposition model to under predict the peak temperatures at this location. However, by weighting this non-linear correction using Equation 4.3, both the maximum and average error can be seen to reduce across the domain.

The results for this case at other inlet velocities can be found in Appendix A.

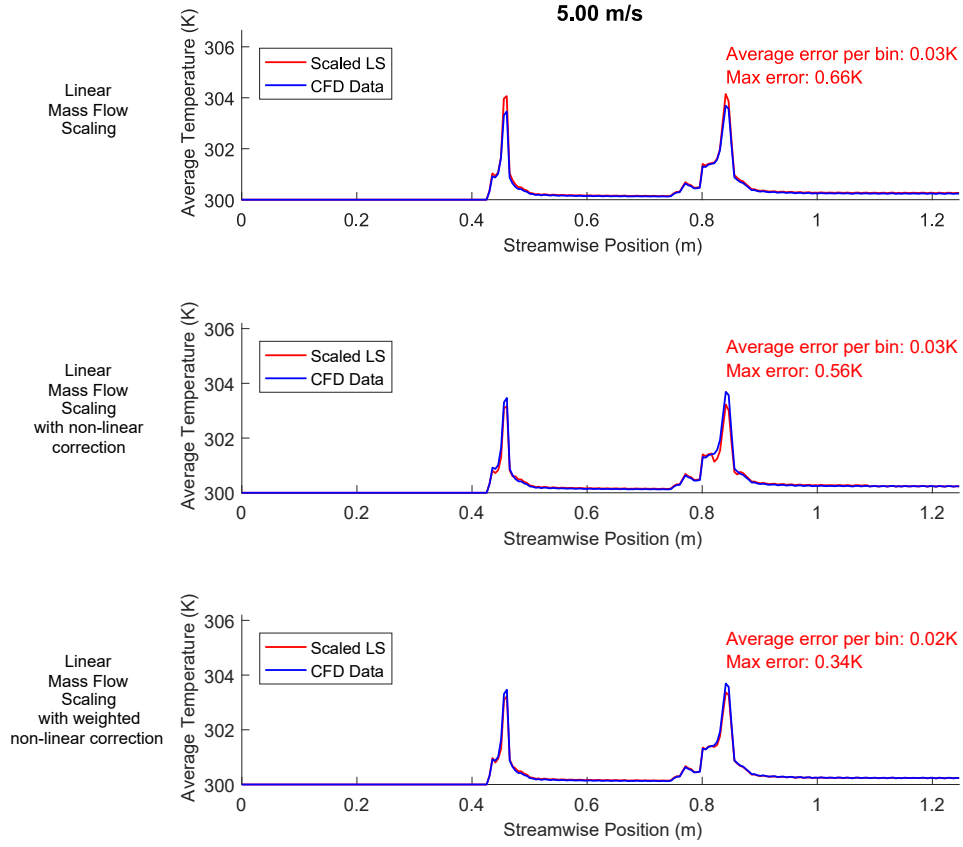


Figure 4.27: Comparison of mass flow linear scaling, mass flow linear scaling with non-linear correction, and mass flow linear scaling with weighted non-linear correction for an inlet velocity of 5 m/s)

4.5 Computation Time

The main advantage of using the linear superposition method is the reduced computational expense compared with running full CFD analyses. The largest benefit to adopting the linear superposition method can be realised where an analysis calls for large numbers of different boundary cases to be investigated for the same geometry.

The computer used to run the present analysis included a Core i7-4470 CPU with 4 physical hyper-threaded cores, running at 3.40 GHz. For the relatively small mesh used in the analysis of the 0.25 scale race car underhood geometry (consisting of 5.6 million tetrahedral elements), the time required for a steady state calculation (using the realisable $k-\epsilon$ turbulence model) to converge was approximately 4 hours, running on 8 threads.

In order to use the linear superposition technique, the temperature influence matrix must be populated using a number of these CFD analyses (equal to the number of heat sources in the system). However, once these initial analyses have been run, the prediction of the temperature field is reduced to a matrix multiplication operation rather than a full 3-D CFD analysis. For the aforementioned underhood model, the time required to predict the temperature field using the linear superposition technique was 1.093 seconds (averaged over 10 predictions) on a single thread.

Given the linear superposition technique is effectively a matrix multiplication operation, this could further be improved through the use of parallelisation. Whilst significant speed up could be realised using parallelisation on CPU cores, acceleration of the calculation could be increased dramatically through the use of GPUs. However, it should be noted that the single threaded performance of the linear superposition method is already far better than an equivalent CFD analysis.

In addition to this, it is noted that the calculations required to populate the temperature influence matrix only vary in their thermal boundary conditions. With the solution to the momentum and continuity equations not changing between these simulations, it is possible to reduce the calculation time of these initial CFD analyses by only resolving the new solution for the energy equation. For the 0.25 scale race car underhood model, this reduces the computation time of a converged solution (solely resolving the energy equation) to approximately 2 minutes.

While this approach could also be used to reduce the time taken to iterate through cases of thermal boundary condition parameter sets, it is noted that the calculation time is two orders of magnitude larger than the equivalent analysis using the linear superposition method.

Chapter 5

Experimental Studies

5.1 Introduction

An experiment was designed to provide validation for the computational results described in Chapter 4. An overview of the experimental setup can be seen in Figure 5.1. The facility comprised an experimental test section, through which air is blown by a centrifugal blower, with capacity for heat addition both at the inlets (via electrical heater meshes) and on the surface of the model (using electrical heaters).

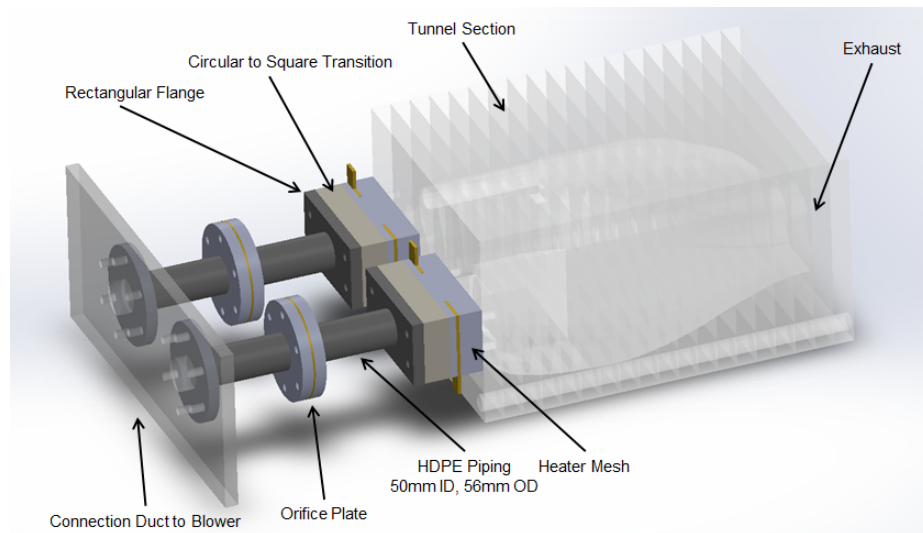


Figure 5.1: Overview of experimental setup (full length of HDPE piping not shown)

5.2 Experimental Test Section

The air flow from the blower is split into two channels which feed the right and left test section inlets respectively (see Figure 5.2). Mass flow through the test section is measured using in-line orifice meters conforming to the European ISO5167-2 specification with one meter installed in each channel. The mass flow through each inlet can be controlled by a ball valve on each channel located upstream of the mass flow meters. The lengths of piping upstream and downstream of the orifice meters (12D and 6D respectively - as specified by the European ISO5167-2 specification) have the additional benefit of helping to condition the flow before it reaches the inlet to the working section.

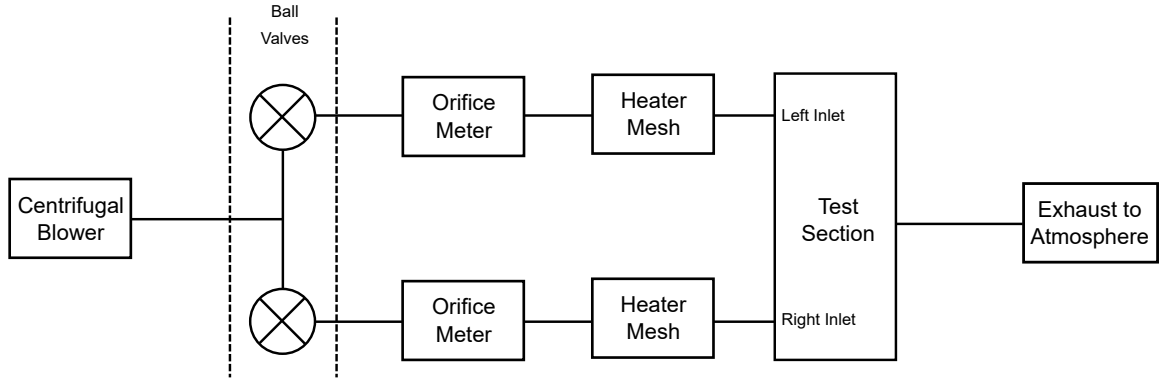


Figure 5.2: Schematic of experimental setup

The experimental setup consists of a 0.25 scale mock up of the rear engine bay of a 2014 Formula One race car. The model was derived from a public domain CAD geometry and was manufactured in the machine shop at the Southwell Laboratory. The model was broken down into 20 individual slices in the axial direction which were then individually cut using a 2D CNC machine from 30 mm slices of Perspex. The internal surfaces were hand polished in order to generate a smooth inner surface. The slices were then carefully stacked and clamped at either end to provide a continuous section (see Figure 5.3).

The test section had two intakes which correspond to the side pod intakes of the Formula 1 race car. Although cooling designs vary between different teams, the majority of cooling strategies, used by race teams, for the engine bay derive the

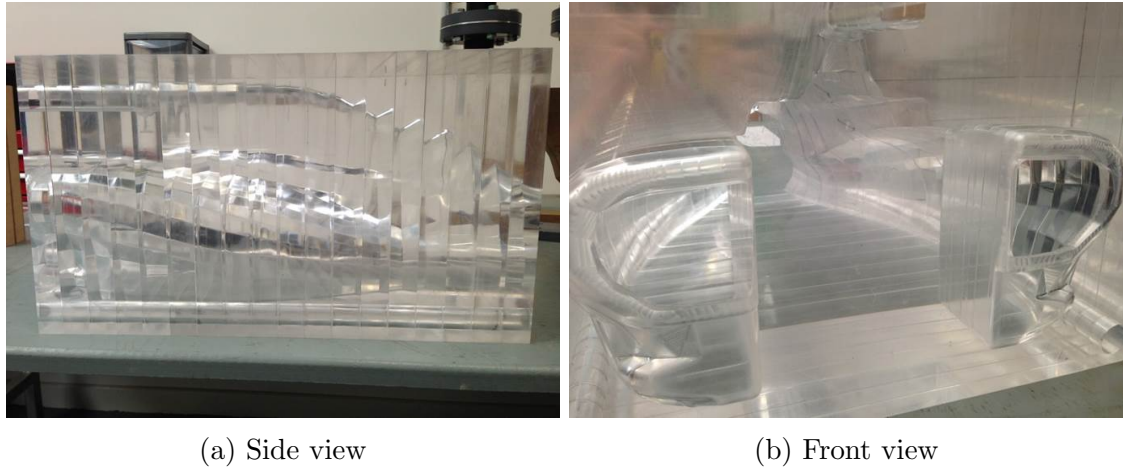


Figure 5.3: Experimental test section

majority of their cooling flow from the side pod intakes. The engine bay top intake is used to feed the internal combustion engine intake and so was not used for the present work.

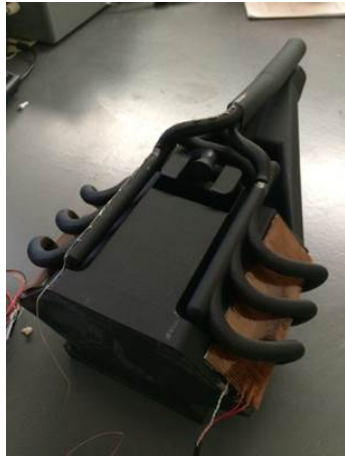
5.3 Internal Assembly

The internal components were created from a model of a generic Formula 1 powertrain conforming approximately to the 2014 FIA regulations. The internal components that were included in the model were the following:

1. Engine assembly
2. Compressor
3. Turbine
4. Gearbox
5. Exhaust and wastegate path

The components modelled were geometrically simplified for ease of manufacture (and for generation of the computational mesh). The simplified assembly was then manufactured as individual parts using fused deposition modelling (FDM) 3D printing. The resulting parts were then joined together to form the overall assembly using epoxy resin. The internal assembly was located in the test section by three threaded helicoil inserts in the base of the assembly (see Figure 5.4b).

The exhaust section of the internal assembly was manufactured separately from Perspex. Perspex was chosen to enable the use of thermochromic liquid crystals for heat transfer coefficient measurements, as the thermal properties of Perspex are well characterised and has previously been shown to provide a good substrate for coating with thermochromic liquid crystals [78].



(a) Internal assembly model



(b) Location of model in test section

Figure 5.4: Internal assembly overview

5.4 Heat Flux Addition

In order to validate the results described in Chapter 4, it was necessary to introduce known heat fluxes into the main gas stream. In the experimental facility, there were two methods of accomplishing this.

5.4.1 Electrical heater pads

The first method was to use electrical heater pads¹ that are mounted on the engine cam covers of the internal assembly (see Figure 5.4a). The electrical heaters consist of an etched foil element encapsulated between two layers of polyimide film (also known as Kapton) giving a heat flux of 15.5 kW/m^2 at a maximum voltage of 115 V. The heat flux output by each of these heaters is controlled by a high voltage variable DC power supply with the heaters connected in parallel to ensure the same potential difference across each heater.

¹Omega Electronics KH-205/10

Table 5.1: Electrical heater pad specifications

	Width (mm)	Height (mm)	Power Density (kW/m ²)	Resistance (Ω)
Left Electrical Heater pad	127	51	15.5	128.6
Right Electrical Heater pad	127	51	15.5	143.2

Although both heater pads had the same specification, the resistances measured independently at room temperature (see Table 5.1) were found to differ significantly. This is likely to have been caused by variance in the manufacturing process. The temperature of each heater pad was monitored using a k-type thermocouple located on the underside of each heater pad and the power supplied to each heater was measured using a digital multimeter and a variable bench power supply.

5.4.2 Electrical heater mesh

The second method of heat flux addition changing the thermal boundary conditions was by way of electrical heater meshes, described by Gillespie et al. [79]. A heater mesh was located in each of the inlets to the experimental section (see Figure 5.2) enabling a step change in the inlet temperature for transient experiments which enables the measurement of heat transfer coefficients. The heat meshes were connected to a high current AC power supply that allowed an abrupt step change in inlet temperature as a result of the short time constant of the mesh wire.

Table 5.2: Heater mesh specification

	Mesh Thickness (microns)	Amperage (A)	Power (W)	Resistance (Ω)	Mean Bulk Gas ΔT (K)
Left Heater	40	97	543	0.058	42
Right Heater	40	97	524	0.056	41

The heater mesh consisted of a fine stainless steel mesh that spans the inlet area which is soldered to two brass buzz bars on each side which act as the connection to the high current power supply. The mesh and buzz bar assembly are then mounted in a laminated plastic (Tufnol) frame which electrically and thermally insulated the conducting mesh wires.

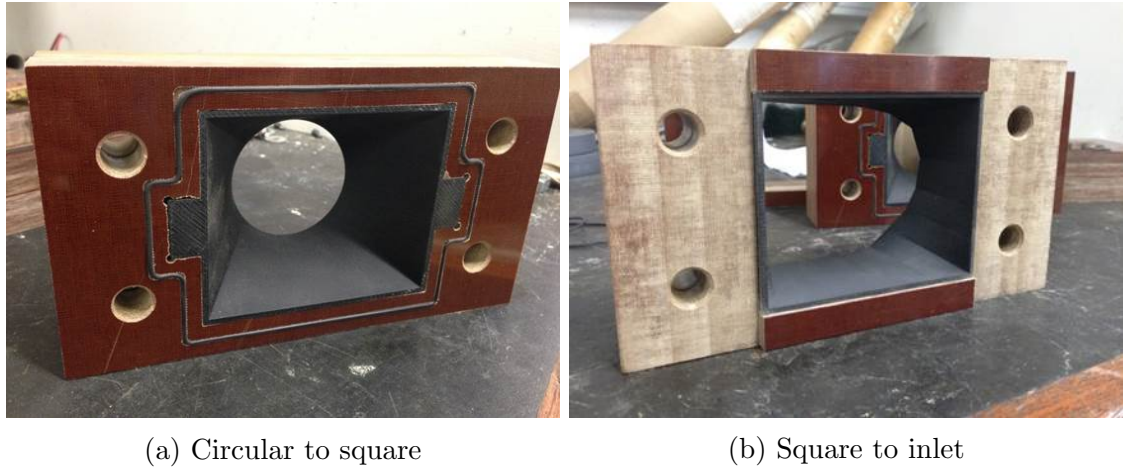


Figure 5.5: Heater mesh transition inserts

There are three distinct geometrical profiles that the air must pass through in each inlet section. The inlet to each heater mesh is circular (necessitated by the connection to the orifice meters), whilst the outlet shares the same geometry as the inlet to the experimental test section. In addition, the profile of the duct is required to be rectangular at the location of the wire mesh. This is due to the high current density through the mesh that requires the heat be dissipated over the entirety of the mesh by a uniform airflow. Without this, the individual strands of the mesh would overheat and fail.

In order to produce smooth transitions between these cross sections, inserts were designed and printed using FDM to fit into the laminated plastic frame which sit immediately upstream and downstream of the heater mesh and buzzbar assembly (see Figure 5.5). These allowed a gentle transition between each of the required cross sectional profiles to prevent separation effects caused by steps at the surface.

5.5 Instrumentation

5.5.1 Point temperature and pressure measurements

The instrumentation on the experimental setup can broadly be separated into two categories based on their use. One set of instrumentation was typically used for steady-state experiments, and another set for transient experiments.

The pressure measurements used to deduce mass flow through each inlet were measured at upstream and downstream locations either side of each orifice meter. The mass flow was controlled by use of two ball valves (shown in Figure 5.2). Pressure measurements were made using pressure transducers² which output an analogue voltage signal that is captured using a 24 bit analogue to digital data logging unit³.

K-type thermocouples were placed at various positions throughout the test section in order to monitor the temperature of various components (such as on the underside of the electrical heater pads). Thermocouples were also installed to measure the air temperature 30 mm downstream of each heater mesh in order to monitor the inlet gas temperature and were located on the vertical mid line at 50% span across the inlet.

Temperature measurements were also performed using a rake of k-type thermocouples at the outlet plane of the test section. Two different rakes were used during testing (shown in Figure 5.6) which both used 0.05 mm diameter wire thermocouples to investigate the temperature of various points in the outlet plane. The coordinates for the thermocouple rakes are shown in Table 5.3.

The data from the thermocouples was logged using several eight-channel 24 bit analogue to digital units⁴ with thermocouple cold junction compensation and amplification, connected via USB. Data was logged from all channels simultaneously at a frequency of 20 Hz.

5.5.2 Measurements using thermochromic liquid crystals

In order to provide validation for the computational analyses performed in Chapter 4, cross-sectional temperature measurements in the domain and heat transfer coefficients on the exhaust part of the internal assembly model (shown in Figure 5.4) were measured.

As heat transfer coefficients are a surface attribute, and are sensibly independent of the thermal boundary conditions of the system, they can provide a useful indication

²Sensortech HCX series pressure transducers

³Picolog ADC-24 data logging unit

⁴Picolog TC-08 data logging unit

Table 5.3: Thermocouple coordinates

Thermocouple Number	Rake 1		Rake 2	
	x (mm)	y (mm)	x (mm)	y (mm)
1	0	117.3	34.3	99.1
2	27.0	106.3	31.4	99.1
3	18.5	106.3	28.5	99.1
4	18.5	93.3	25.6	99.1
5	18.5	84.3	22.7	99.1
6	18.5	73.3	19.8	99.1
7	10.0	84.3	16.9	99.1
8	n/a	n/a	14.0	99.1
9	-10.0	84.3	-14.0	99.1
10	-18.5	73.3	-16.9	99.1
11	-18.5	84.3	-19.8	99.1
12	-18.5	93.3	-22.7	99.1
13	-18.5	106.3	-25.6	99.1
14	-27.0	106.3	-28.5	99.1
15	n/a	n/a	-31.4	99.1
16	n/a	n/a	-34.3	99.1

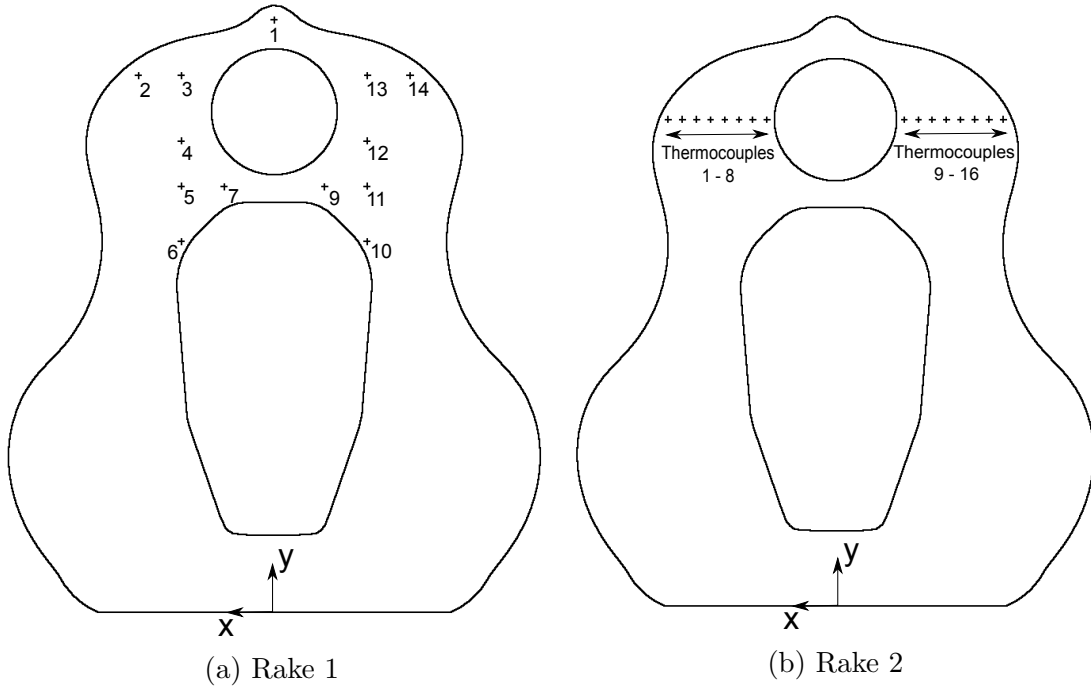
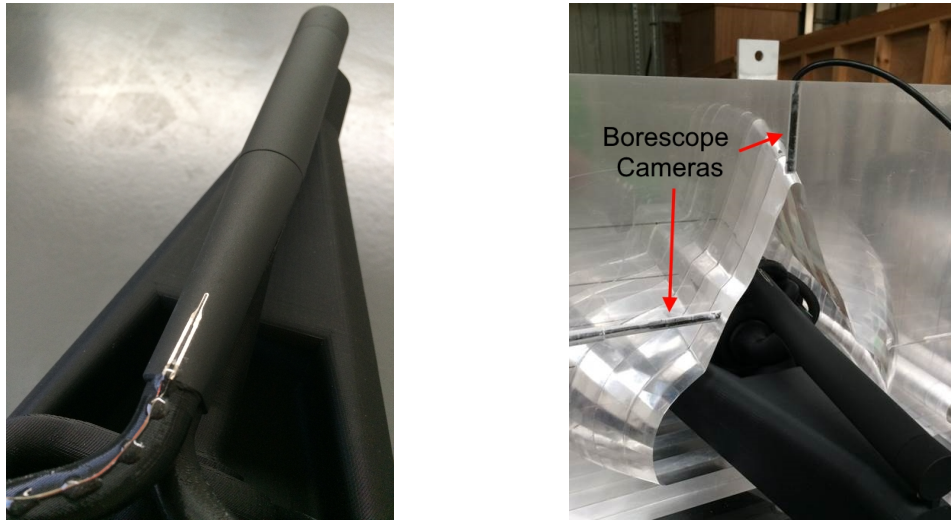


Figure 5.6: Thermocouple rakes (as viewed from rear of car)

of the velocity field in the domain which can then be used to evaluate agreement with computational models.

In addition to the measurement of the heat transfer coefficients on the exhaust part, thermochromic liquid crystals were also used to investigate the cross-sectional temperature profile within the experimental geometry. To accomplish this, a thermochromic liquid crystal coated mesh was suspended in the gas flow path.

5.5.2.1 Thermochromic liquid crystals



(a) Location of surface thermocouple on exhaust component (b) Example of instrumented section with borescope cameras

Figure 5.7: Instrumentation in experimental test section

The use of thermochromic liquid crystals for transient heat transfer experiments is well established [80]. The use of thermochromic liquid crystals allows wide area temperature measurements with a high spatial resolution. Thermochromic liquid crystal temperature measurements are achieved by viewing light reflected back from the liquid crystal coating in the visible spectrum so that conventional imaging equipment can be used.

The advantages of using thermochromic liquid crystal temperature measurements over other spatial methods are the increased spatial resolution and the ability to record data simultaneously for different geometrical points. In addition, the use of

thermochromic liquid crystal temperature measurements are less invasive than alternative methods as the majority of the associated equipment can be located outside the test section.

A thermochromic liquid crystal coating changes colour depending on its temperature. The recorded image colour for the thermochromic liquid crystals is typically post-processed using the green intensity observed throughout the test. Although the use of other parameters (such as hue value) has been used by other researchers [81] [82], recent work has shown that green intensity provides increased sensitivity for weak signals [83].

In order to process the data from the thermochromic liquid crystal coating, it is necessary to record the RGB values of the thermochromic liquid crystals throughout each experiment. The images were collected using narrow cylindrical profile (7 mm diameter), fixed focal length borescopes⁵ embedded in the walls of the experimental test section (see Figure 5.7b).

5.5.2.2 Thermochromic liquid crystal calibration

The liquid crystals are calibrated in situ, before the experiments, with the same lighting used in the heat transfer experiments. Using different gas temperatures, images of the surface of interest are recorded under the same conditions as the experiment with both the green intensity values (calculated from the imaging equipment) and surface temperature (from the surface thermocouple, seen in Figure 5.7a) recorded simultaneously. Hence, by relating the logged green intensity values to the measurements of the surface thermocouple, it is possible to calibrate the observed green intensity values to the temperature of the liquid crystal coating.

5.5.2.3 Imaging equipment

Low cost, narrow profile borescopes were used to record the colour response of the coating throughout the experiment. Given the fixed focal length of these cameras and the inability to vary the distance between the lens and surface of interest (by virtue of the necessity of embedding them in the wall to provide minimal disruption to the flow field), multiple cameras were used to image as much of the surface of interest as

⁵Supereyes N005 USB borescopes

possible. The use of multiple cameras was facilitated by the low cost of each unit, which are orders of magnitude cheaper than similar cameras used for heat transfer research.

The cameras are connected to a data logging PC using a USB interface and are capable of recording at VGA resolution (640 x 480 pixels). Commercial image capture software is used to capture multiple video streams simultaneously. In order to allow smooth video capture using all six cameras simultaneously, the resolution of each video stream is reduced to 320 x 240 pixels in order to avoid exceeding the bandwidth of the USB controller on the workstation motherboard.

5.5.2.4 Image mosaicing

Due to the proximity of each borescope to the imaged surface and the fixed focal length of the borescopes, each individual camera had a limited field of view. In order to provide a more complete view of the surface being imaged, image processing techniques were leveraged to provide a means of mosaicing the image from individual borescopes.

Any two images of the same planar surface can be defined by the relationship $x' = Hx$, where x' denotes a point in the image plane, x denotes a point in the base plane, and H is defined as the homography matrix (shown in Equation 5.1) [84].

$$\begin{pmatrix} x' \\ y' \\ 1 \end{pmatrix} = \begin{bmatrix} h_{11} & h_{12} & h_{13} \\ h_{21} & h_{22} & h_{23} \\ h_{31} & h_{32} & h_{33} \end{bmatrix} \begin{pmatrix} x \\ y \\ 1 \end{pmatrix} \quad (5.1)$$

As the value of h_{33} can be assumed to be unity, it is possible to define the homography matrix using 4 corresponding points of similarity. By selecting the view plane of a certain borescope as the base plane, the homography matrix enables images from other borescopes to be transformed into base plane by selecting 4 points of similarity in areas of overlapping imagery. Once each of the images have been transformed into the same image plane, the same points of similarity can then be used to mosaic the images together to form a single image.

5.5.2.5 Cross sectional temperature measurements

In order to measure the temperature across the cross section of the geometry, a thermochromic liquid crystal coated nylon mesh was suspended in the gas flow path. The mesh was located 120 mm upstream of the outlet plane. The use of a nylon mesh to measure cross-sectional temperature profile has previously been reported by Wang, Ireland, and Kohler [81].

The porosity of the mesh was measured to be 0.895 and the corresponding total pressure drop across the mesh was reported as 0.22 by Murray [85]. A mesh with a high porosity was chosen in order to minimise the obstruction to the gas flow path.

The nylon mesh was coated with a 1°C narrow band thermochromic liquid crystal with an activation temperature of 25°C . The mesh was imaged using the borescopes described in Section 5.5.2.3. The resulting imagery was then processed and mosaiced using a MATLAB tool written by the author, assisted by Murray [85]. An example of the resulting mosaiced image of the mesh is shown in Figure 5.8.

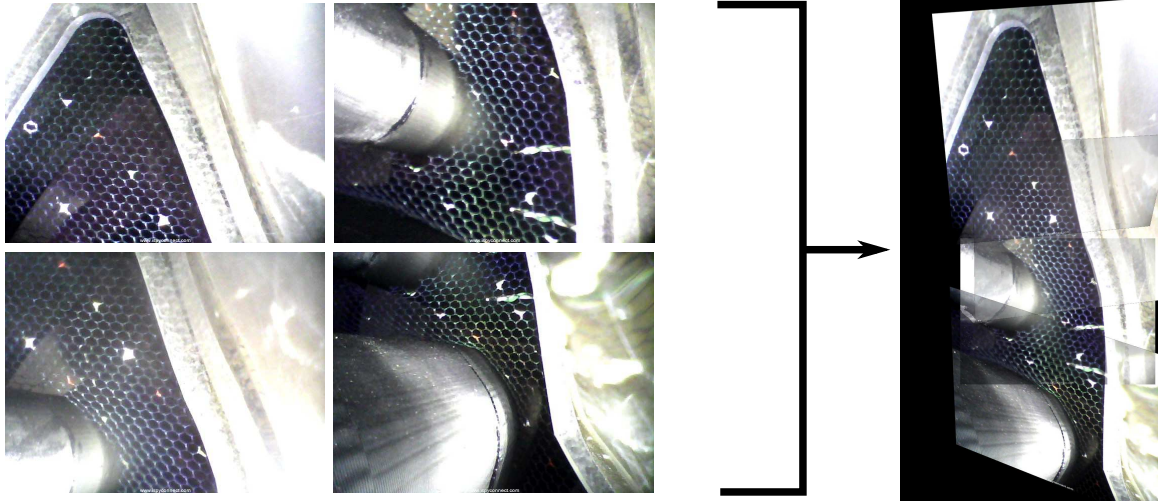


Figure 5.8: Example of mosaiced mesh imagery

5.5.2.6 Heat transfer coefficient calculation

The use of thermochromic liquid crystals (and the surface thermocouple for calibration) allows the accurate measurement of the model exhaust part surface temperature over a significant area during each transient experiment. The exhaust part was coated

with a 1°C narrow band thermochromic liquid crystal, with an activation temperature of 35°C. The surface temperature measurement allows the calculation of the local heat transfer coefficients (as defined in Equation 5.2).

$$h = \frac{q}{T_g - T_s} \quad (5.2)$$

Each experiment begins with a step change in the inlet air temperature, which allows the local heat transfer coefficient to be determined at each pixel by assuming that the local substrate can be considered as a semi-infinite plate (such that the thickness of the surface of interest is sufficient that the transient response is the same as that of a semi-infinite substrate). The analytical solution to the Fourier equation (described by Clifford et al [86]) in Equation 5.3 is then solved for the parameter β , and thus for the local heat transfer coefficient, h :

$$T_s = T_0 + (T_g - T_0)[1 - \exp(\beta^2) \operatorname{erfc}(\beta)] \quad (5.3)$$

where:

$$\beta = \frac{h\sqrt{t}}{\sqrt{\rho c_p k}}$$

$\sqrt{\rho c_p k}$ = Thermal product

Once the calibration of the liquid crystal has been completed (using the method described in Section 5.5.2.2), the green intensity values at each timestep of the experiment is predicted for different values of the convective heat transfer coefficient for known model thermal product, initial model temperature, and gas temperature using Equation 5.3. These theoretical green intensity profiles are then collected in a 2-D array (with indices corresponding to heat transfer coefficient and time).

The video data from the experiment is spatially discretised using the green intensity value at each pixel. Using the data from each frame of the video, this results in green intensity profile data for each pixel for the duration of the experiment. Each experimental profile can then be matched with a theoretical green intensity profile (using a least squares cost function) in the aforementioned 2-D array in order to estimate the local heat transfer coefficient at each pixel in the video data [87].

5.6 Experimental Results

5.6.1 Steady state temperature measurements

5.6.1.1 Temperature rake measurements

The first set of experiments examined the steady state temperature measured in the outlet plane of the test section using the thermocouple rakes described in Section 5.5.1. This procedure was used to examine if the linear scaling, described in Chapter 3, could be verified with the experimental data.

To test the steady state performance of the test geometry, the electrical heater pads (described in Section 5.4) were used to introduce a known heat flow into the main air flow within the test section. The resulting temperature field was then investigated at the outlet plane. The experiment was then repeated many times with the two rake setups described in Section 5.5.1 and with varying amount of power supplied to the heater pads. In order to account for changes in the experimental environment between tests, the temperature observed by each of the thermocouples (T_i) was non-dimensionalised using the following formula described in Equation 5.4.

$$\Phi_i = \frac{T_i - T_{in}}{\frac{Q}{\dot{m}c_p}} \quad (5.4)$$

The uncertainty in the measurement of the dimensionless temperature parameter is investigated in Appendix C.

The first set of experiments investigated the accuracy of superposing two sets of thermal boundary conditions to predict the temperature field that would arise with the corresponding combined thermal boundary conditions. In these experiments, the air velocity was set to 2.5 m/s at the inlet plane to the test section and the heat flux supplied was varied for three test conditions (see Table 5.4). In each case, the heater pads were given one minute to reach their steady state temperature and the data was recorded for two minutes thereafter. The resulting data were then averaged before plotting.

Table 5.4: Superposition of individual cases - Full power - Rake 1

Experiment Name	Left Heater Pad Power (W)	Right Heater Pad Power (W)	Total Heat Flux Addition (W)
Run 1	0	83.6	83.6
Run 2	93.5	0	93.5
Run 3	88.8	76.4	168.2

Table 5.4 shows that the heater pads were first run individually at a set voltage (the differences between the heat flux supplied is due to variations in resistance between the heater pads) in Runs 1 and 2, for comparison to results from Run 3. Using the theory set out in Chapter 3, it should be possible to superpose the temperature measurements from Runs 1 and 2 in order to achieve the same temperature field at the outlet plane as observed using the thermal boundary conditions set out in Run 3.

In the case of Run 3, both heaters were powered simultaneously, the heater pads were connected in parallel to supply the same voltage across each of the heater pads. As the current draw was below the maximum that the power supply was able to source, this ensured consistency between each test as the heater pads would supply the same heat flux when running simultaneously as individually. The tests were then repeated with the voltage across the heater pads halved, which reduced the heat flux supplied by a factor of four (see Table 5.5).

Table 5.5: Superposition of individual cases - Reduced power - Rake 1

Experiment Name	Left Heater Pad Power (W)	Right Heater Pad Power (W)	Total Heat Flux Addition (W)
Run 1	0	20.9	20.9
Run 2	23.4	0	23.4
Run 3	22.2	19.6	41.8

5.6.1.2 Rake 1 results and discussion

Figures 5.9 and 5.10 show the measured thermocouple data with the thermal boundary conditions described in Tables 5.4 and 5.5 respectively. It can be seen that the

agreement between the superposed cases and the explicit boundary conditions is very good with minimal discrepancies.

A key point of agreement is the similarity in temperature profile measured across the thermocouples. As the dimensionless temperature parameter, Φ , is scaled linearly with the heat flux addition, this shows the temperature field scales linearly with heat flux addition as predicted in Chapter 3.

Figures 5.11 and 5.12 show the thermal influence coefficients (as defined in Equation 3.4) for the left and right electrical heater pads when run independently. The full and reduced power conditions indicate the thermal boundary conditions prescribed in Tables 5.4 and 5.5 respectively. It can be seen that there is minimal lateral mixing of the flow through the working section with the left electrical heater creating a raised temperature profile on the left hand side (as viewed from the front) and vice versa for the right electrical heater. This is likely due to the lateral blockage caused by the internal assembly model.

The thermal influence coefficients provide a quantitative method of indicating whether a thermal field resulting from a set of thermal boundary conditions can be linearly scaled with changes to the boundary conditions. From a qualitative standpoint, the temperature profiles observed by the thermocouples agree well with each other. Generally, the results between the full power and reduced power conditions show good quantitative agreement for the majority of point measurements which indicates good agreement with the linear superposition principle described in Chapter 3.

5.6.1.3 Rake 2 results and discussion

Figures 5.13 and 5.14 show the dimensionless temperature (as defined in Equation 5.4) for the point measurements as specified for Rake 2 (see Table 5.3). Figures 5.13 and 5.14 show a comparison between linearly superposed Run 1 and Run 2 cases (as defined in Tables 5.4 and 5.5) and the combined explicit boundary conditions (defined as Run 3). This is repeated for both full power and reduced power as is the case in Section 5.6.1.2.

The results follow a similar pattern to the results in Section 5.6.1.2 with there being very good agreement between the linear superposed case and the explicit boundary

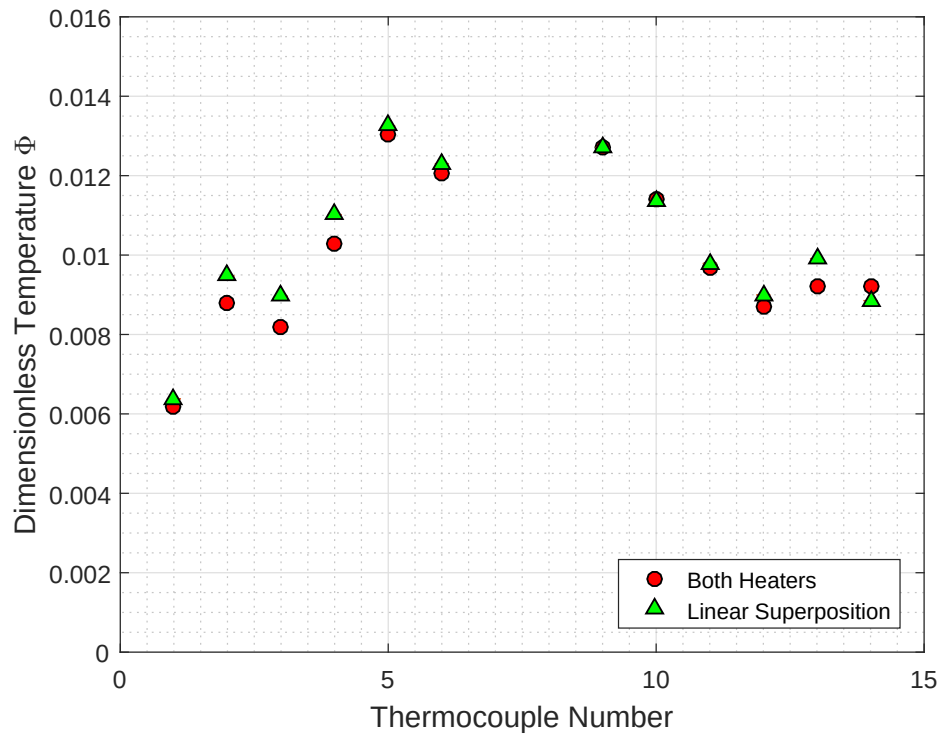


Figure 5.9: Superposition of individual cases - Full power - Rake 1

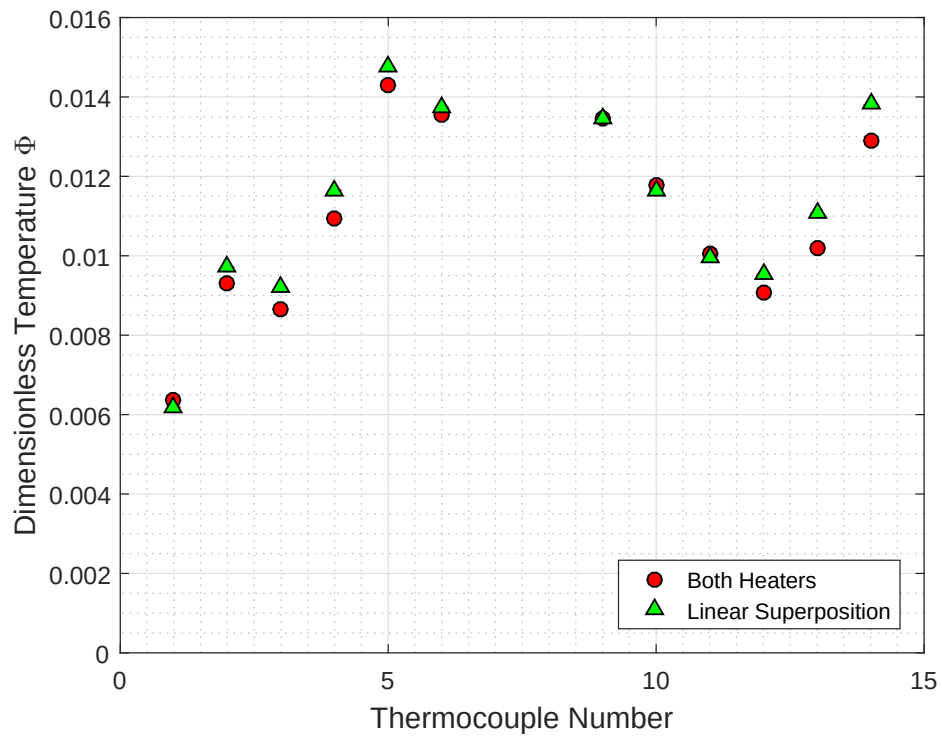


Figure 5.10: Superposition of individual cases - Reduced power - Rake 1

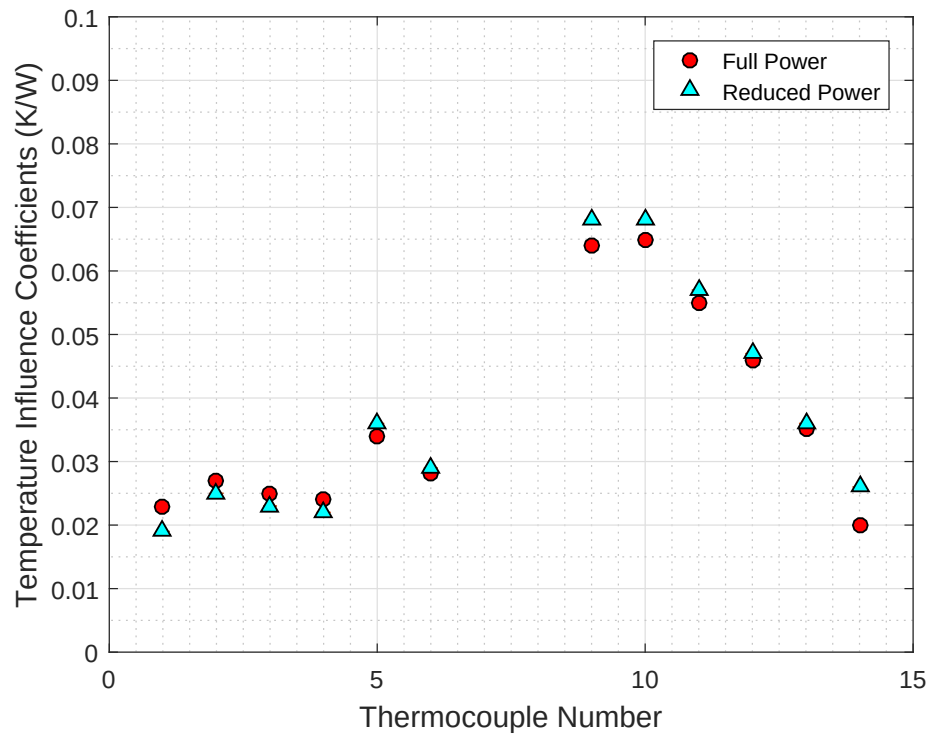


Figure 5.11: Thermal influence coefficients - Left heater - Rake 1

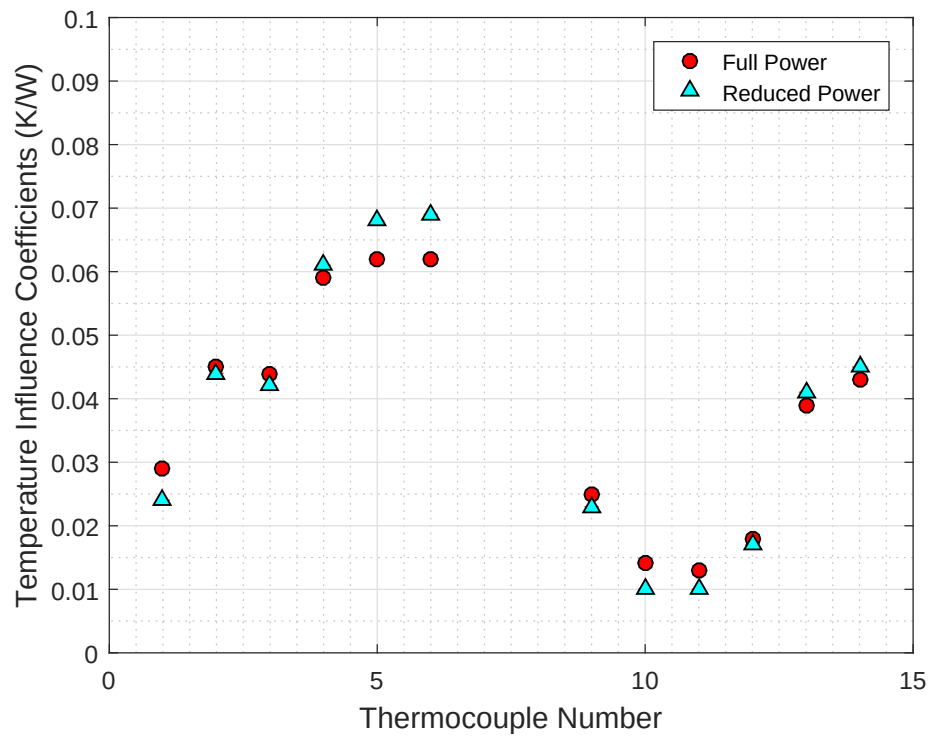


Figure 5.12: Thermal influence coefficients - Right heater - Rake 1

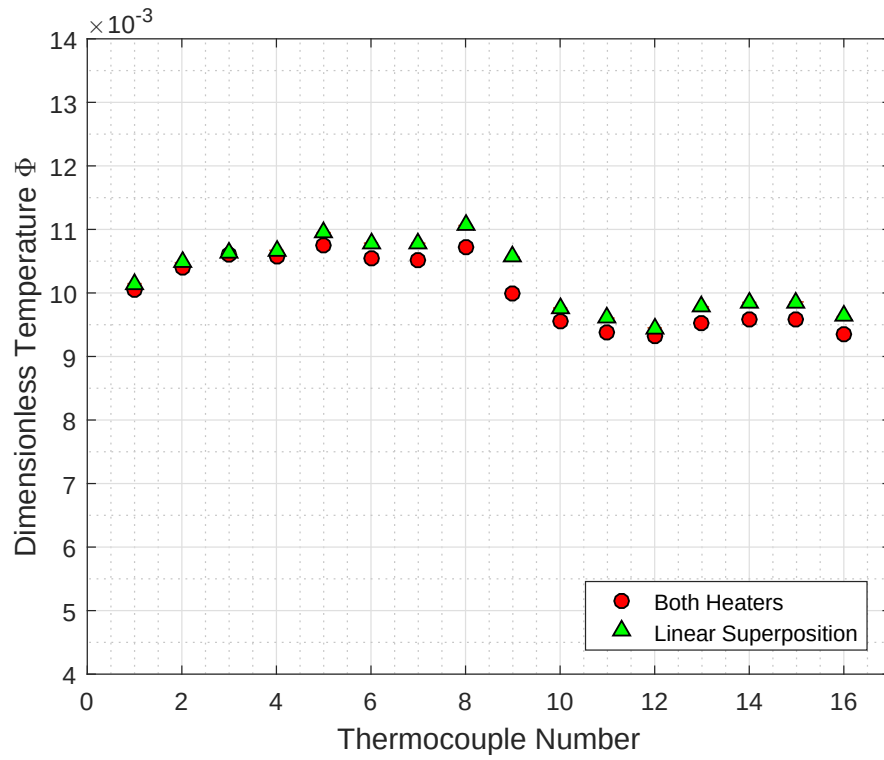


Figure 5.13: Superposition of individual cases - Full power - Rake 2

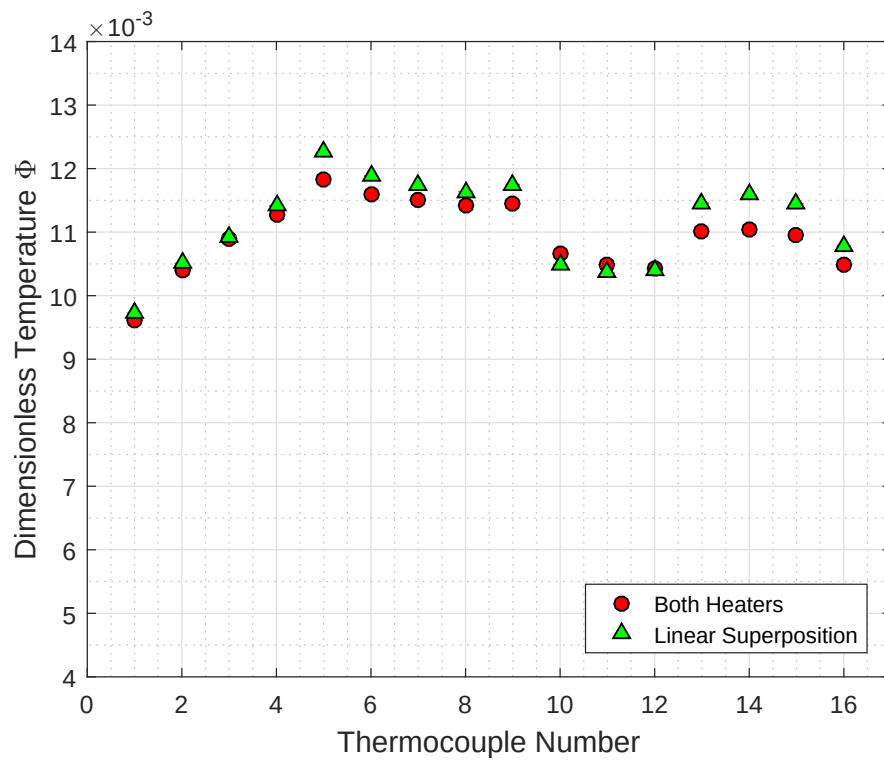


Figure 5.14: Superposition of individual cases - Reduced power - Rake 2

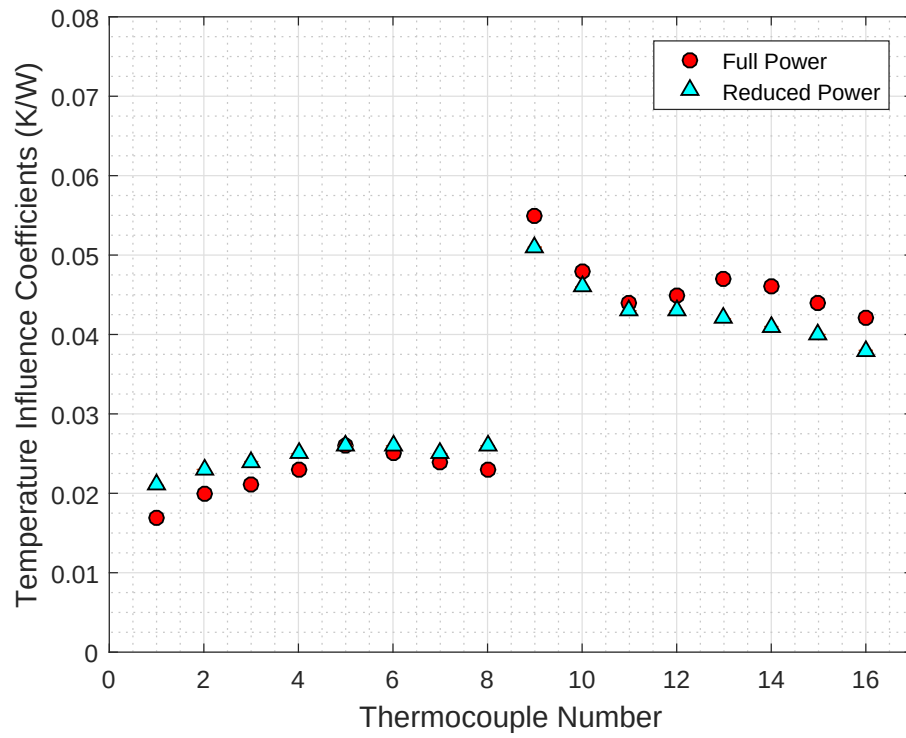


Figure 5.15: Thermal influence coefficients - Left heater - Rake 2

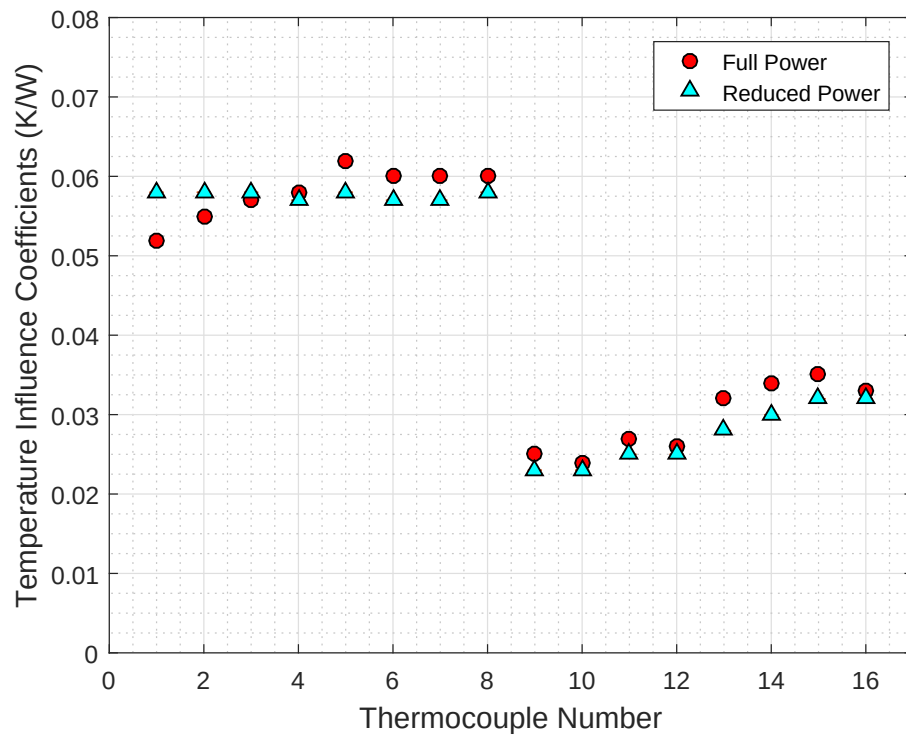


Figure 5.16: Thermal influence coefficients - Right heater - Rake 2

condition case. The similarity in temperature profiles shown in Figures 5.13 and 5.14 also show the temperature field scaling linearly with the heat flux addition.

Figures 5.15 and 5.16 show the thermal influence coefficients (as defined in Equation 3.4) for the left and right electrical heater pads run independently while the full and reduced power states indicate the conditions set out in Tables 5.4 and 5.5. These provide a quantitative assessment on whether the linear superposition principle is applicable at different thermal boundary conditions.

As expected from the data in Section 5.6.1.2, the results in Figures 5.15 and 5.16 shows there is minimal mixing of the airflow from the left and right inlets occurring within the domain. This is shown by the clear difference in temperature influence coefficients when the left and right electrical heater pads are independently active.

As shown in Figures 5.11 and 5.12, the temperature influence coefficients show fairly good agreement between the different thermal boundary conditions with slight discrepancies likely to be attributed to changes in the ambient conditions between each test.

5.6.1.4 Cross-sectional temperature results

Figure 5.17 shows the temperature influence coefficients on a nylon mesh suspended in the gas flow path, 120mm upstream of the outlet plane, for varying inlet velocities. The same thermal boundary conditions as in Run 3 (see Table 5.4) were used.

The nylon mesh was imaged using the method described in Section 5.5.2.3. The resulting images were then transformed and mosaiced together using the technique outlined in Section 5.5.2.4. In order to remove background noise from the images, an edge detection algorithm was used to produce a edge-based binary filter of the image. As the edges of the nylon mesh are well defined, this edge-based binary filter could then be morphologically dilated and used as a binary mask to only display data from the mesh.

The colour space of the resulting filtered image was then processed to calculate the temperature of the mesh, from which the temperature influence coefficients could then be calculated. This was then compared against numerical simulations with the same boundary conditions using the model described in Section 4.1.

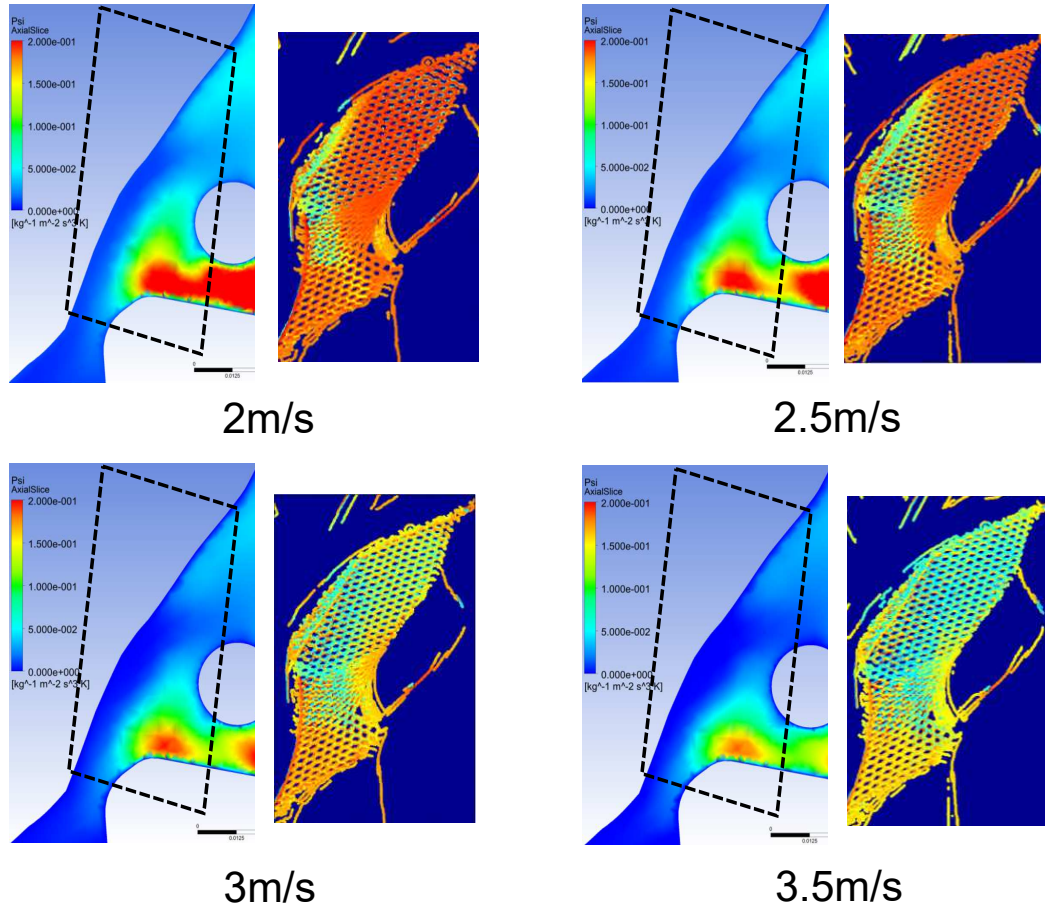


Figure 5.17: Comparison between CFD (left) and mosaiced experimental values (right) of cross-sectional temperature influence coefficients at varying inlet velocities

The agreement between the experiments and the numerical simulations appears to be poor at first observation. However, the mesh will only return useful information when the thermochromic liquid crystal is activated. In regions where the temperature of the mesh is above or below the bandwidth of the thermochromic liquid crystal, the temperature will be reported as 26°C or 25°C respectively.

In the region between the minimum and maximum temperature influence coefficients, some agreement can be observed between the numerical simulations and the experimental data. This is evident in the region close to the wall, where the area of reduced temperature influence coefficient can be seen to spread with increasing inlet velocity. Similarly, the areas of increased temperature influence coefficient at the top right and bottom right of the domain shown in Figure 5.17 can be observed in both the experimental and computational data.

5.6.2 Transient heat transfer experiments

5.6.2.1 Thermochromic liquid crystal experimental method

Heat transfer coefficient measurements were made on the exhaust section of the internal components. The exhaust section was manufactured from Perspex (unlike the rest of the internal assembly which was made using a rapid prototyping FDM machine) and painted matte black in order to give a good optical substrate for the thermochromic liquid crystal coating. The exhaust section was coated using a narrow band crystal (as described in Section 5.5.2.1) with a 35°C activation temperature and 1°C bandwidth. A surface thermocouple was also bonded to the surface of the exhaust part (see Figure 5.7a) in order to both calibrate the thermochromic liquid crystal activation temperature and to provide a method of calculating the point heat transfer coefficient.

The video recording and the activation of the thermocouple logger must be accurately synchronised in order to allow the correct computation of the heat transfer coefficients. In order to aid this synchronisation, a light (visible in the video imagery) was used to indicate power being supplied to the heater meshes.

The results from the liquid crystal coating were compared against a point heat transfer coefficient measurement made using the surface thermocouple located on the top side of the exhaust. The surface thermocouple is visible in the video imagery which allows a direct comparison between the local heat transfer coefficient calculated using the surface thermocouple and the local heat transfer coefficient at the adjacent pixel, calculated using the thermochromic liquid crystals. An analysis of the uncertainty in the local heat transfer coefficient measurement is shown in Appendix C.

5.6.2.2 Validation of thermochromic liquid crystal results

The first set of experiments were designed to test the green intensity-temperature calibration of the thermochromic liquid crystals and the robustness of the synchronisation method (outlined in Section 5.6.2.1) and to assess whether the heat transfer coefficients calculated using the thermochromic liquid crystal method were sensible and accurate. In order to assess the accuracy of results, the image data from the USB

borescopes was post-processed using an in-house software tool [88] to provide estimations for the spatial heat transfer coefficients on the exhaust part. These were then compared to the heat transfer coefficients calculated from the surface thermocouple (shown in Figure 5.7a) using Equation 5.3.

Table 5.6: Mean ΔT , and heat transfer coefficients calculated using surface thermocouple

Inlet Velocity (ms^{-1})	Mean $\Delta T(K)$	Mean Heat Transfer Coefficient ($Wm^{-2}K^{-1}$)
2	84	21
3	54	29
4	29	53

Table 5.6 shows the mean heat transfer coefficient (calculated using the surface thermocouple) for the duration of the experiment along with the mean step change in gas temperature (generated by the heater meshes located over each inlet) observed during the experiment.

The values can be compared with the data in Figures 5.18, 5.19, and 5.20 which show the local heat transfer coefficients on the exhaust surface calculated using the thermochromic liquid crystals. It can be seen that the heat transfer coefficient, calculated using both techniques, at the location of the surface thermocouple are very similar with agreement within 5%.

5.6.2.3 Computational model validation using heat transfer coefficients

In order to provide validation for the computational model (described in Section 4.1), an additional experiment was performed. The inlet velocity was set to 2 m/s for the duration of the test. In order to generate the step change in gas temperature, a current is passed through each of the electrical heater meshes that span the inlets to the working section. The power was supplied using a high current AC power supply that delivered 144 A at 3.5 V across the mesh which resulted in a step change in gas temperature of 56.34 K across the heater mesh.

Figures 5.21 and 5.22 show the heat transfer coefficients on the side and top of the

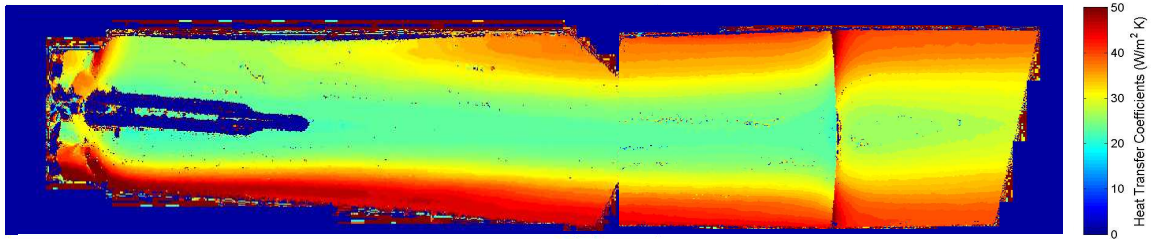


Figure 5.18: Measured heat transfer coefficients on exhaust top surface with 2 m/s inlet velocity (see Figure 5.7a)

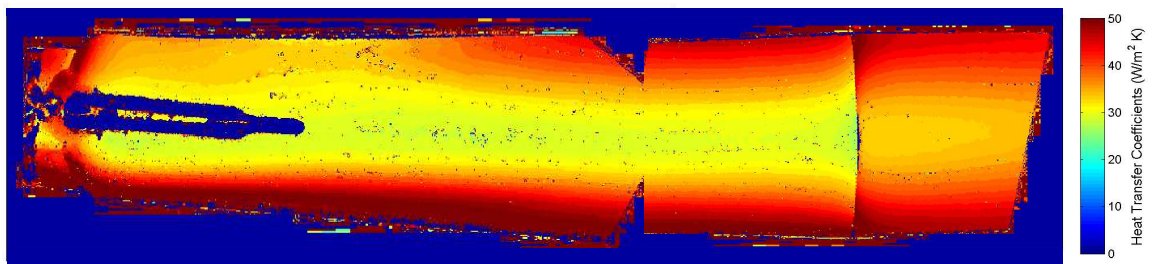


Figure 5.19: Measured heat transfer coefficients on exhaust top surface with 3 m/s inlet velocity (see Figure 5.7a)

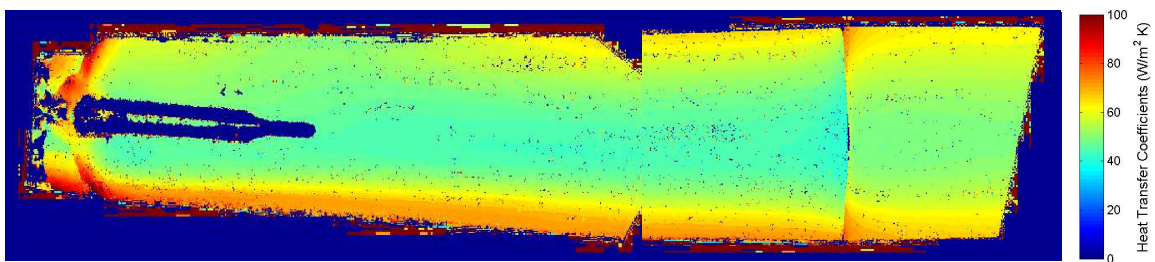


Figure 5.20: Measured heat transfer coefficients on exhaust top surface with 4 m/s inlet velocity (see Figure 5.7a)

exhaust part respectively enabling a comparison between the experiments and computational model (described in Section 4.1). The experiments and computational simulations show clear similarities in both the magnitude and distribution of the heat transfer coefficients.

There are some clear differences between the computational simulations and experiment in the area where the separate parts of physical model have been joined. This causes a disruption in the smoothness of the model surface which is likely to cause some disruptions to the flow field in this area. Disruption in the measured heat transfer coefficient due to physical discontinuities can also be observed in Figures 5.18, 5.19, and 5.20. These discontinuities are located where the internal assembly has been joined together and can be seen in Figure 5.7a.

The results support the interpretation of the flow field discussed in Sections 5.6.1.2 and 5.6.1.3 as it can be observed that the heat transfer coefficients are noticeably higher on the sides of the exhaust part (see Figure 5.21) than the top section (see Figure 5.22). This is caused by the lack of mixing between the mass flows from the left and right inlets.

The amount of curvature present was not deemed to be sufficient to invalidate the assumption of 1-D conduction. However, the accuracy of the data is likely to be reduced in areas where curvature is present. This is due to a combination of a reduction in spatial resolution (as each pixel then covers more surface area) and the increase in distance from the surface of interest to the lens of the borescope. As the focal length of each borescope is fixed, any increase in distance between the surface of interest and the borescope lens causes a large reduction in imaging quality.

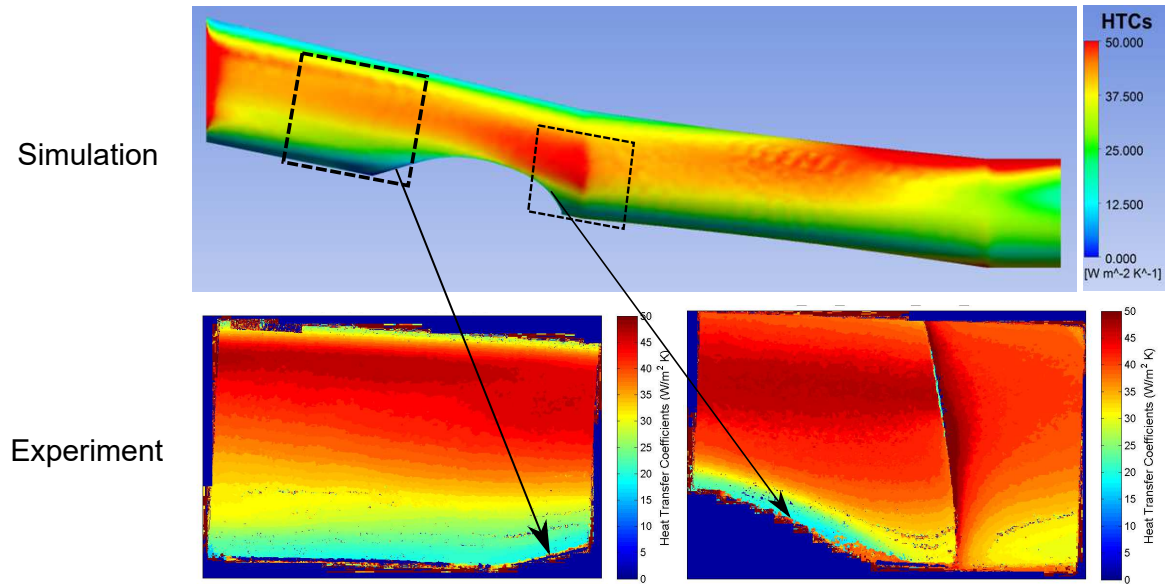


Figure 5.21: Heat transfer coefficients on exhaust side for 2 m/s inlet velocity

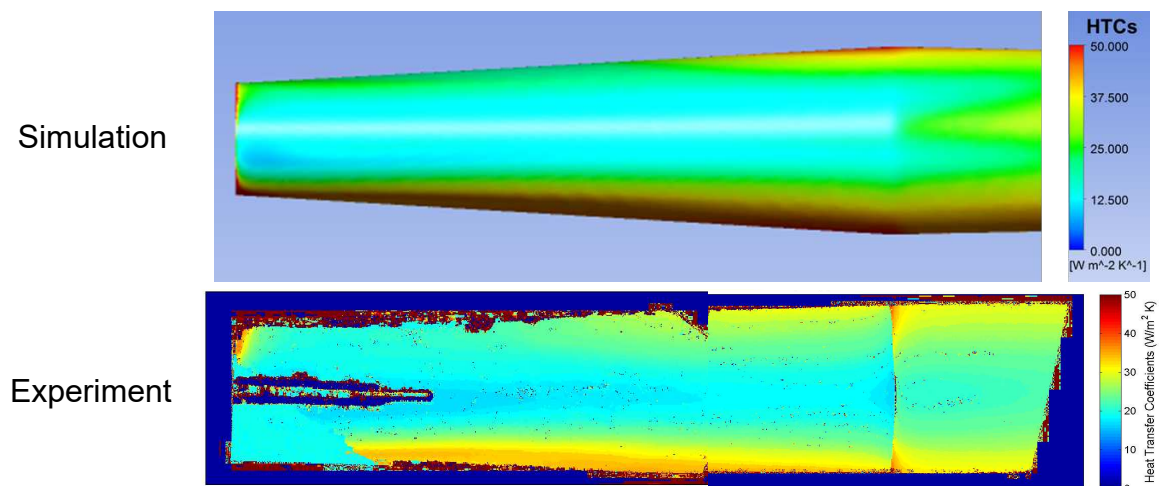


Figure 5.22: Heat transfer coefficients on exhaust top surface for 2 m/s inlet velocity
(see Figure 5.7a)

5.7 Summary of Linear Superposition Model Development

The work in Chapters 3, 4, and 5 detailed the development and implementation of a linear superposition method for 3-D temperature field investigations in complex geometries. The method was demonstrated for both varying thermal and inlet speed boundary conditions, with a non-linear correction applied to account for Reynolds number dependency for varying flow velocities.

The method was used to computationally investigate the temperature field in two domains with good agreement shown between the linear superposition method and CFD simulations for varying thermal boundary conditions. Agreement between the linear superposition method and CFD simulations for varying inlet boundary condition was shown to improve with the use of the non-linear correction described in Section 3.6. The advantage in computational speed over the use of CFD simulations (considered to be the industry standard) was illustrated.

An experimental facility, based on the quarter scale Formula 1 underhood geometry, was built in order to validate the linear superposition model. Temperature rakes were used to measure steady state temperatures at the outlet plane with the results showing good agreement with the linear superposition model. Thermochromic liquid crystals were used to investigate the heat transfer coefficients on the exhaust of the internal assembly model, with the results showing good agreement with CFD simulations. A novel method of imaging thermochromic liquid crystals coated surfaces using multiple low-cost borescopes was demonstrated, with the resulting image streams mosaiced together to form a single holistic image.

Chapter 6

Application of Thin Film Gauge Technology to In-Cylinder Heat Flux Measurements

6.1 Introduction

This chapter details the work undertaken to obtain heat flux measurements on the inner surface of an internal combustion engine cylinder. In order to obtain the measurements, a novel temperature sensor was developed, capable of surviving the harsh conditions inside the cylinder during the combustion process. The sensor was designed to be used in a pre-existing instrumentation fitting in the clearance volume of a single cylinder test bench and was capable of obtaining crank resolved temperature measurements.

The use of thin film technology is well suited to obtaining high bandwidth temperature measurements. Using knowledge of the substrate thermal properties, the temperature measurements could then be processed to obtain heat flux data. The results were then compared to empirical models for in-cylinder heat flux, developed by Woschni [5] and Hohenberg [6].

6.2 Traditional Heat Flux Measurement Techniques

There are many methods of measuring heat flux, as summarised in the seminal work by Schultz and Jones [35]. The majority of techniques reported in the aforementioned

literature were pioneered for gas turbine or hypersonic flow research where a combination of high magnitude heat fluxes, a requirement for fast sensor response, and hostile environments can be found.

The measurement of heat fluxes within a internal combustion engine cylinder presents a significant challenge as the gas temperature in the cylinder can reach values of approximately 2500 K with in-cylinder pressure peaking at values of up to 250 bar during the combustion process. These conditions result in heat fluxes of up to 3 MW/m² into the cylinder wall. In addition, the bandwidth of the sensor was specified to provide crank resolved measurement at 15,000 RPM, which necessitates a sensor response frequency of approximately 100 kHz.

Given these requirements, thin film gauge technology was judged to be the most suitable method of heat flux measurement. For ease of installation, the proposed probe was designed to fit into the mount of an existing pressure transducer¹.

6.3 Thin Film Gauge Technology

Thin film gauges are widely used to measure transient heat transfer. The technique was initially developed to investigate heat transfer in short duration hypersonic shock tubes [37] and has been extensively developed and refined at the Osney Thermo-Fluids Laboratory, Oxford. A summary of this development is comprehensively described by Jones [89]. A more recent summary of thin film gauge development is described by Piccini et al. [90].

A thin film gauge consists of a thin electrically conducting platinum resistor (typically between 0.1 μm and 1.0 μm thickness [35]) which is located on the gas swept surface of an electrically insulating layer or substrate. Current is supplied to the resistor using tracks of gold or copper whose resistance can be considered negligible compared to the sensing element.

A constant current source is used to supply current to the gauge whose resistance varies linearly with temperature (see Equation 6.1). The constant current source is designed to pass a very low current (between 10 mA and 20 mA) so as not to cause any ohmic heating within the gauge. As the thin film gauge undergoes a temperature

¹Kistler 6054BR pressure transducer

change, this causes a proportional change in voltage across the gauge. The reference conditions, R_0 and T_0 , are normally measured at 0°C.

$$R = R_0 [1 + \alpha_{tc}(T_s - T_0)] \quad (6.1)$$

Many different substrates have been used in the construction of thin film gauges. Traditionally, a vitreous enamel insulating layer deposited on a metal base was used [91], but was then superseded by the use of machinable glass ceramic (such as Corning MACOR) [92] which made the metal base unnecessary. For both the machinable glass and vitreous enamel substrates, the platinum resistance sensor and connecting tracks are usually painted by hand using a metallo-organic ink. The gauge is then fired to leave behind electrically conducting tracks. A flexible polyimide (Kapton) substrate has also been used extensively for low temperature applications, with the gauges formed using chemical etching and photoresist [93]. This allows for much finer tracks to be created, which enables denser arrays of thin film gauges.

Machinable glass ceramic is typically used for high temperature applications (such as those described in Section 6.2). However, machinable glass has poor mechanical properties, thus making it unsuitable for applications where the gauge is subjected to high mechanical loads. Machinable glass based sensors have previously been used inside internal combustion engines [70], but the latter report highlighted robustness issues caused by the mechanical strength of the machinable glass substrate.

With this in mind, two different approaches were considered as alternatives to the high temperature machinable glass and low temperature Kapton substrates. The first approach used the two layer gauge (described by Doorly and Oldfield [91]) which comprises an electrically insulating layer deposited on top of a metal base. This approach increases the robustness of the gauge for high temperature, high pressure applications, relative to the machinable glass based sensors.

The second approach explored the use of pure ceramic materials, in particular alumina and fused quartz. Ceramic materials have better mechanical properties than machinable glass which makes them more suitable to internal combustion engine applications, where pressure loads and stresses caused by thermal cycling can be significant.

6.3.1 Calculation of heat flux from surface temperature

The platinum resistance sensor is used to measure the surface temperature of the substrate and the signal processed to obtain the local heat flux. As described in Section 6.3, both single and two-layer gauges are investigated. Calculation of the heat flux signal from the surface temperature signal for both system requires the solution of the 1-D Fourier equation for a semi-infinite boundary condition. For the two-layer system, two solutions of the 1-D Fourier equation are required for both the insulating and base metal layers.

6.3.1.1 Single layer heat transfer gauge

For a single layer gauge system, the thin film is deposited on an insulating substrate which is typically considered to be infinite in the positive x direction (see Figure 6.1).

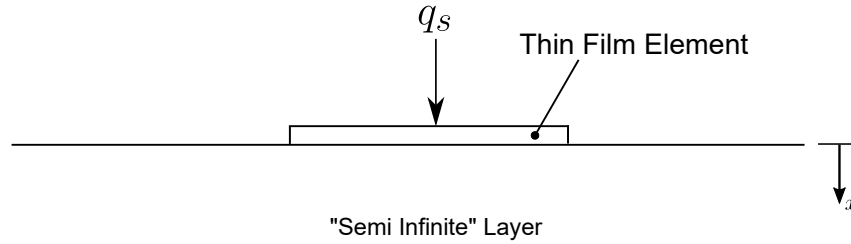


Figure 6.1: Thin film gauge on single layer substrate with semi-infinite metal layer

The heat flux into the gauge can be calculated from the solution to the 1-D Fourier equation, shown in Equation 6.2. This assumes that the heat transfer within the system can be considered as 1-D (ie. that the lateral condition in the y and z directions is negligible) and that the thermal pulse does not reach the limits of the substrate. Schultz and Jones [35] report that the thermal penetration depth can be estimated from $4\sqrt{\alpha t}$, which should be significantly less than the thickness of the substrate for the semi-infinite solution to be applicable.

$$\frac{\partial T_1}{\partial t} = \alpha_1 \frac{\partial^2 T_1}{\partial x^2} \quad (6.2)$$

The boundary conditions are specified as follows:

$$-k \frac{\partial T_1}{\partial x} = q_s \quad \text{at } x = 0 \quad (6.3)$$

$$T = 0 \quad \text{at } x = \infty \quad (6.4)$$

We then take the Laplace transform of Equation 6.2 to arrive at Equation 6.5, where $\overline{T}_s$ and $\overline{q}_s$ denote the Laplace transform of the surface temperature and surface heat flux respectively and s denotes the Laplace transform variable.

$$\overline{T}_s = \frac{1}{\sqrt{\rho c k}} \frac{\overline{q}_s}{s} \quad (6.5)$$

Inverting the Laplace transform then gives the relationship between surface temperature and surface heat flux (as shown in Equation 6.6).

$$T_s = \frac{1}{\sqrt{\pi} \sqrt{\rho c k}} \int_0^t \frac{q_s(\tau) d\tau}{(t - \tau)} \quad (6.6)$$

6.3.1.2 Two-layer heat transfer gauge

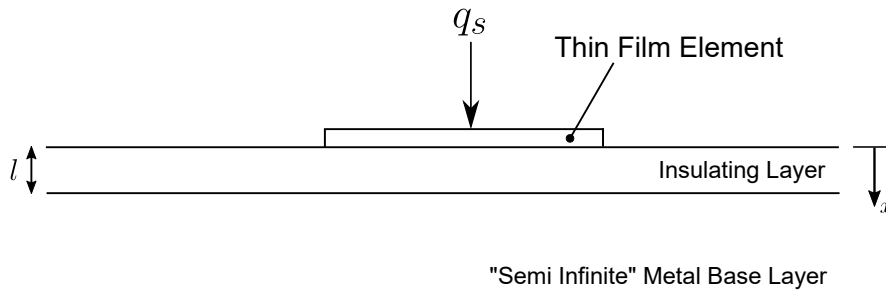


Figure 6.2: Thin film gauge on two-layered substrate with semi-infinite metal layer

The two-layer gauge comprises a thin film platinum resistance sensor applied to the gas swept side of an insulating layer deposited over a thick substrate which is typically metallic (shown in Figure 6.2). This allows the problem to be thought of as two

coupled solutions to the 1-D Fourier equation (see Equations 6.7 and 6.8), subject to the boundary conditions described in Equations 6.9, 6.10, 6.11, and 6.12. Solutions to Equations 6.7 and 6.8, subject to the aforementioned boundary conditions, have been described by Doorly and Oldfield [94].

$$\frac{\partial T_1}{\partial t} = \alpha_1 \frac{\partial^2 T_1}{\partial x^2} \quad (6.7)$$

$$\frac{\partial T_2}{\partial t} = \alpha_2 \frac{\partial^2 T_2}{\partial x^2} \quad (6.8)$$

The boundary conditions are specified as follows:

$$-k \frac{\partial T_1}{\partial x} = q_s \quad \text{at } x = 0 \quad (6.9)$$

$$k \frac{\partial T_1}{\partial x} = k \frac{\partial T_2}{\partial x} \quad \text{at } x = l \quad (6.10)$$

$$T_1 = T_2 \quad \text{at } x = l \quad (6.11)$$

$$T_2 = 0 \quad \text{at } x = \infty \quad (6.12)$$

The solution of Equations 6.7 and 6.8, using Laplace transforms, yields an expression which relates the surface temperature to the surface heat flux (see Equation 6.13).

$$T_s(t) = \frac{2q_s}{\sqrt{\rho_1 c_1 k_1}} \left[\sqrt{\frac{t}{\pi}} + \sum_{n=1}^{\infty} 2A^n \left\{ \sqrt{\frac{t}{\pi}} \exp\left(-\frac{k_s^2}{4t}\right) - \frac{k_s}{2} \operatorname{erfc}\left(\frac{k_s}{2\sqrt{t}}\right) \right\} \right] \quad (6.13)$$

where:

$$A = \frac{\sqrt{\rho_1 c_1 k_1} - \sqrt{\rho_2 c_2 k_2}}{\sqrt{\rho_1 c_1 k_1} + \sqrt{\rho_2 c_2 k_2}}$$

$$k_s = \frac{2ln}{\sqrt{\alpha_1}} = 2n\sqrt{\rho_1 c_1 k_1} \left(\frac{l}{k_1} \right)$$

Equation 6.13 can then be used to calculate the heat flux into the thin film gauge from the measured surface temperature signal and known values of the substrate thermal properties.

6.3.2 Impulse response processing in transient heat transfer experiments

The equation relating heat flux and surface temperature (see Equation 6.2) has traditionally been solved using numerical methods as outlined by Schultz and Jones [35]. However, this method is computationally expensive and scales poorly as the amount of data increases. In addition to this, solutions for the surface heat flux often involves the computation of derivative terms which can be complicated by the presence of noise and artifacts in experimental data.

A method that makes use of fast Fourier transforms was suggested by Oldfield [95] and is considered here. The fast Fourier transform method treats the gauge as a black box linear time invariant (LTI) system, where the properties of the system do not change with time. By treating the inputs and outputs (heat flux and temperature or vice versa) to the system as a discrete series, the method applies discrete deconvolution to calculate the filter impulse response of the system for a known input and output. This impulse response filter is then used to filter a discrete input series to recover the output of the system using discrete convolution.

For a LTI system that has surface temperature, $T(t)$, and surface heat flux, $q(t)$, as its input and output respectively, the relationship between the input and output of the system can be calculated by convolving the impulse response function, $h_{T-q}(t)$, with the surface temperature, $T(t)$, as shown in Equation 6.14.

$$q(t) = h_{T-q}(t) * T(t) = \int_{-\infty}^{\infty} h_{T-q}(\tau) T(t - \tau) d\tau \quad (6.14)$$

Equation 6.14 is difficult to solve in the continuous-time domain due to singularities at the origin [35], which can be overcome by evaluating Equation 6.14 in the discrete-time domain. In addition, input signals to the system (either surface temperature or heat flux) are often obtained from experimental data and already a discrete series. For data acquired with a sampling period T_s , the surface temperature is given by Equation 6.15.

$$T[n] = T(nT_s) \quad (6.15)$$

In the discrete-time domain, Equation 6.14 becomes the discrete convolution sum, shown in Equation 6.16.

$$q[n] = h_{T-q}[n] * T[n] = \sum_{k=-\infty}^{\infty} h_{T-q}[k]T[n-k] = \sum_{k=-\infty}^{\infty} h_{T-q}[n-k]T[k] \quad (6.16)$$

In order to calculate the discrete impulse response function, $h_{T-q}[n]$, a pair of non-singular signals, $q_1(t)$ and $T_1(t)$, are required (known as the basis functions) [95]. These basis functions are used to determine the impulse response of heat flux to an impulse in surface temperature. If these basis functions are sampled to create a discrete series using the same sample period, T_s , as the experimental data, they can be considered to share the same impulse response function $h_{T-q}[n]$ as in Equation 6.17.

$$q_1[n] = h_{T-q}[n] * T_1[n] \quad (6.17)$$

The most efficient way of calculating the discrete impulse response function $h_{T-q}[n]$ is to deconvolve the signal using a discrete delta function ($\delta[n] = 1, 0, 0, 0, \dots$). Using z-transforms, it is possible to rewrite Equation 6.17 using a multiplication sum as in Equation 6.18.

$$Q_1(z) = H_{T-q}(z)T_1(z) \quad \text{or} \quad H_{T-q}(z) = \frac{Q_1(z)}{T_1(z)} \quad (6.18)$$

By definition, convolving the impulse response function with an impulse function will return the impulse response function. Therefore, in the discrete domain, it is possible to recover the impulse function by using Equation 6.19.

$$H_{T-q}(z) = H_{T-q}(z)\Delta(z) = \frac{Q_1(z)}{T_1(z)}\Delta(z) \quad (6.19)$$

Thus, the impulse response function $h_{T-q}[n]$ can be obtained by digitally filtering the impulse function $\delta[n]$ using a rational transfer function with numerator $q_1[n]$ and denominator $T_1[n]$. This procedure is computationally expensive and can be time

consuming for a large number of points. However, once the impulse response function $h_{T-q}[n]$ has been calculated for a given system, it can be stored and reused for any input to the same system. The impulse response function needs to be recalculated when there are changes in the length of the processed data, the sampling frequency of the data, or when there are changes to the underlying system behaviour (such as changes in the materials used).

6.4 Calibration of Thin Film Gauges

The thin film gauges are calibrated using the isothermal water bath rig at the Osney Thermo-Fluids laboratory.

The calibration rig consists of a thermostatically controlled water bath controlled by an instrumentation system². The system controls the temperature of the water bath and reads data from various K-type thermocouples situated in the water bath. The PXI system allows for the calibration of up to 96 gauges simultaneously.

The thin film gauge to be calibrated is placed in a water proof bag and is submerged in the water bath using weights. A K-type thermocouple is also placed within the bag (normally attached to the specimen) and is connected to the PXI system. The water temperature is then incremented in 5°C steps with resistance measurements across the specimen recorded using a 22 bit digital multimeter. The resistance measurement is recorded by the PXI system when the temperature conditions are considered to be isothermal. The conditions are adjudged to be isothermal when the measurements from the thermocouples in the system (2 internal waterbath thermocouples and any thermocouples attributed to the samples being calibrated) are kept within 0.3 K of each other for a set amount of time (normally set to be 5 minutes). The above steps are often then repeated to reduce the uncertainty in the resulting calibration.

Figure 6.3 shows a typical calibration curve generated by the isothermal water bath rig. From Equation 6.1, the relationship between temperature and gauge resistance is expected to be linear. From the data in Figure 6.3, it is possible to calculate the α_{tc} and R_0 parameters (see Equation 6.1) for each gauge using linear regression. The reference temperature, T_0 , is nominally taken to be 0°C.

²National Instruments PXI system

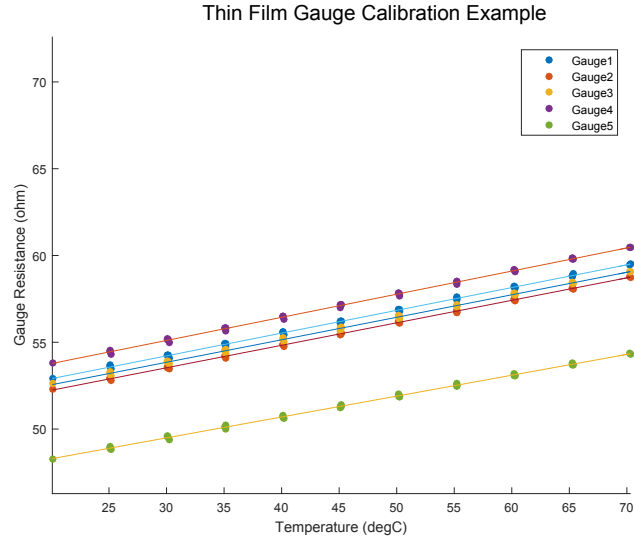


Figure 6.3: Example calibration of thin film gauges

In addition to the calculation of the α_{tc} and R_0 parameters, the 95% confidence intervals are also calculated using a t-based confidence test. The confidence intervals can be used as a rough estimate of the gauge quality,

6.5 Base Design for Heat Flux Sensor

6.5.1 Base geometry

As outlined in Section 6.1, the proposed sensor was designed to fit into an existing instrumentation fitting in order to avoid the need to make extensive changes to existing experimental test facilities. The instrumentation fitting was designed for an in-cylinder pressure transducer³ located in the cylinder head. The sensor was designed to access the clearance volume of the cylinder, so as to avoid any problems caused by moving parts.

Basing the core geometry of the probe around the mount for the pressure transducer constrained several aspects of the design from the beginning. These constraints included the use of an M5 x 0.5 thread and the diameter (2 mm) of the probe tip, which restricted the overall size of the sensor.

For the multi-layer thin film gauge system, the gauge substrate was manufactured

³Kistler 6054BR piezo-electric pressure transducer

from Inconel 625. Inconel 625 is widely used in the aerospace industry as it offers the advantage of high mechanical strength at elevated temperatures. In addition, it has previously been used for multi-layer gauge systems [91]. For the single-layer thin film gauge design, a ceramic core was used as the substrate for the thin film gauges. This was then fitted within a stainless steel collet using an interference fit to keep it in place.

6.5.2 Cable routing

The difficulty of cable routing is one of the key challenges to using thin film gauges in hostile environments. The use of direct metal laser sintering (DMLS) rapid prototyping to manufacture the base geometry of the sensor (Inconel 625 for the multi-layer system and stainless steel for the single-layer system) enabled the manufacture of intricate geometries. This allowed novel methods of routing cables from thin film gauge sensors to be trialled as described below.

Although the platinum sensing element and gold tracks were robust enough to withstand temperature up to 900 K, the solder interface with the wiring was expected to fail at approximately 600 K. As the combustion gases can reach temperature of up to 2500 K, a thermally shielded area was needed for the solder interface between the gold tracks and wiring. Wires can then be attached at this interface and are routed to the rear of the probe using through holes which span the length of the sensor (see Figure 6.4).

The solution for this was to use a gasket in order to seal the area between the combustion gases and the solder joint (see Figure 6.4). The gasket was designed to achieve an interference fit with the conical head of the probe so as to provide a tight seal between the mount in the cylinder head and the probe. With this in mind, the gasket was in contact with the gold leads on the conical head of the probe and so it was important that the gasket was electrically insulating. In addition to this, the pressure in the cylinder can reach up to 150 bar, which places requirements on the gasket's mechanical strength.

The initial materials considered for the gasket were machinable glass and alumina. However, when tightening the sensor in the instrumentation mount to the required

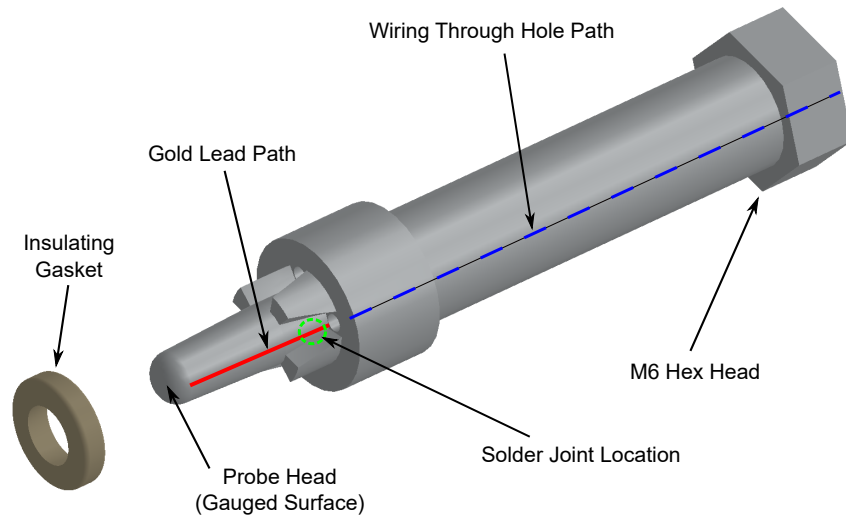


Figure 6.4: Overview of thin film gauge design

torque, the compression forces resulted in the fracture of both materials. Therefore, it was decided that a more ductile material was required. The solution was to use an aluminium gasket, coated in an oxide layer to provide electrical insulation.

6.6 Design and Manufacture of a Two-Layer Heat Flux Sensor

6.6.1 Decision factors for coating choice

Inconel 625 was chosen as the material for the two-layer heat flux sensor base layer following the work of Doorly and Oldfield [91], who developed a two-layer sensor for use in short duration wind tunnels. An electrically insulating coating is required to electrically isolate the sensor and its connecting tracks from the base metal. The coating was required to:

1. provide an electrically insulating layer
2. provide a smooth and non-porous surface to enable platinum and gold to be deposited
3. have suitable mechanical and thermal properties that would enable the sensor to survive in an operating IC engine

The base metal coating needed to be electrically insulating in order to prevent a short circuit between the sensing circuit and the base metal.

The second requirement was that the coating should provide a smooth and non-porous surface. This is important at the initial deployment of the conducting elements, as the platinum and gold are applied using a metallo-organic ink. The surface roughness of the gauge can cause the electrical tracks to be uneven and discontinuous, which affects the consistency and reliability of the resulting gauge. The coating should also be non-porous to prevent the electrical tracks from leeching through the coating, causing a short circuit with the base metal.

The third consideration relates to the material properties of the coating, both mechanical and thermal. As the coating would be directly subjected to the conditions inside the engine cylinder (see Section 6.2), it is important that the mechanical hardness of the coating is sufficient to resist significant abrasion during engine operation. In addition, the thermal properties of the coating was also considered. As the liquid platinum and gold were fired at 680°C to deposit conducting tracks, the coating was required not to deform below this temperature.

Another important material property to consider is the coefficient of thermal expansion of the coating which should be similar to the base metal. When the coefficients of thermal expansion are matched, this reduces the stresses caused by differential thermal expansion of the base metal and coating. The last consideration is the coating thermal conductivity, which influences the differential thermal expansion of the base metal and coating by affecting the temperature field.

6.6.2 Insulator coating materials

Traditionally, Inconel 625 has been paired with a vitreous enamel coating [91] [94] with the platinum gauge elements and gold leads applied to the gas swept surface. In addition to vitreous enamel⁴, a plasma spray applied coating of alumina-zirconia⁵ was also investigated.

⁴Supplied by Ellison Coating Systems

⁵Supplied by Zircotec Ltd.

The advantage of using vitreous enamel is that the combination of vitreous enamel and Inconel 625 has previously been used in research in Oxford [91] and the method of preparation is well established. Vitreous enamel is generally regarded as a suitable coating for thin film gauge applications due its low porosity, low surface roughness and good electrical insulation capabilities. The disadvantage of vitreous enamel is its current lack of use for research purposes. It was difficult to source a supplier who could accurately coat the required area in a repeatable manner to the desired specification. This was particularly an issue with the wiring through holes which were often blocked in the coating process (described in Section 6.5.2). This was compounded by the difficulty in post-machining the parts to clear the blockage, as the diameter to length ratio (combined with the material hardness) resulted in several drill bit failures.

The advantages of using an alumina-zirconia coating was the accurate characterisation of the material properties and the controllable and repeatable method of application. The disadvantages of using the alumina-zirconia coating was the relatively high porosity (approximately 9%) and high surface roughness of the finish which presented problems during the deployment of the platinum and gold. The cost of applying the alumina-zirconia coating was also high compared to the vitreous enamel coating.

The material properties for the relevant materials are listed in Table 6.1.

Table 6.1: Properties for thin film gauge materials

Material	Coefficient of Thermal Expansion (K^{-1})	Thermal Conductivity ($Wm^{-1}K^{-1}$)
Inconel 625	126×10^{-7}	15
Vitreous Enamel	125×10^{-7}	0.90
Alumina-Zirconia	100×10^{-7}	15

6.6.3 Initial observations on platinum and gold deployment

Two Inconel 625 discs were manufactured and coated for initial trials. The discs were cut from 2 mm Inconel 625 plate with diameter 50 mm. The material properties of Inconel 625 plate was assumed to be the same as DMLS Inconel 625.

Each disc was instrumented with five platinum thin film gauges (see Figure 6.5) using the process described in Section 6.1. The aim was to determine the suitability of each coating to the deployment of thin film gauges. The coated discs with painted gauges are shown in Figure 6.6.

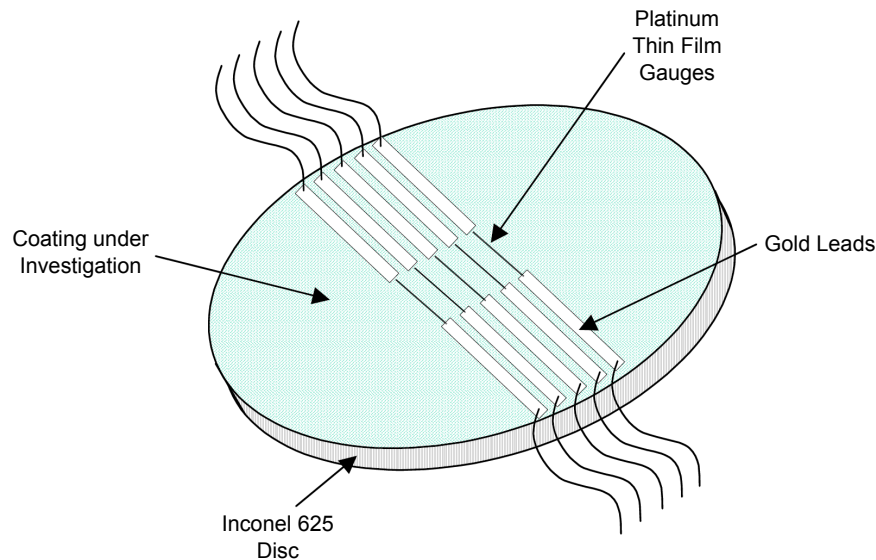


Figure 6.5: Coated Inconel 625 disc for coating investigation [96]

It was observed that the platinum deposition on the vitreous enamel coating was not repeatable and the gauges suffered several failures after the firing. Two problems were observed during the firing process which are thought to cause these manufacturing issues. Firstly, the vitreous enamel was observed to undergo slight liquefaction during the firing process, causing the coating to flow. This was evidenced by the coating moving on to the sides of the disc due to gravitational effects. This movement of the surface underneath the liquid platinum, combined with the surface tension of the liquid platinum, was thought to cause the liquid platinum to separate during the early stages of the firing process, causing gaps in the resulting platinum resistor.

As a result of the above, several stages of rework were required for each gauge deployed on the vitreous enamel surface, which consists of reapplying the liquid platinum and gold on top of the existing conducting tracks and re-firing. As this resulted in a greater thickness of platinum, this reduced the resistance of the gauge. The reduced

resistance results in a reduction the response time and signal amplitude from the gauge.

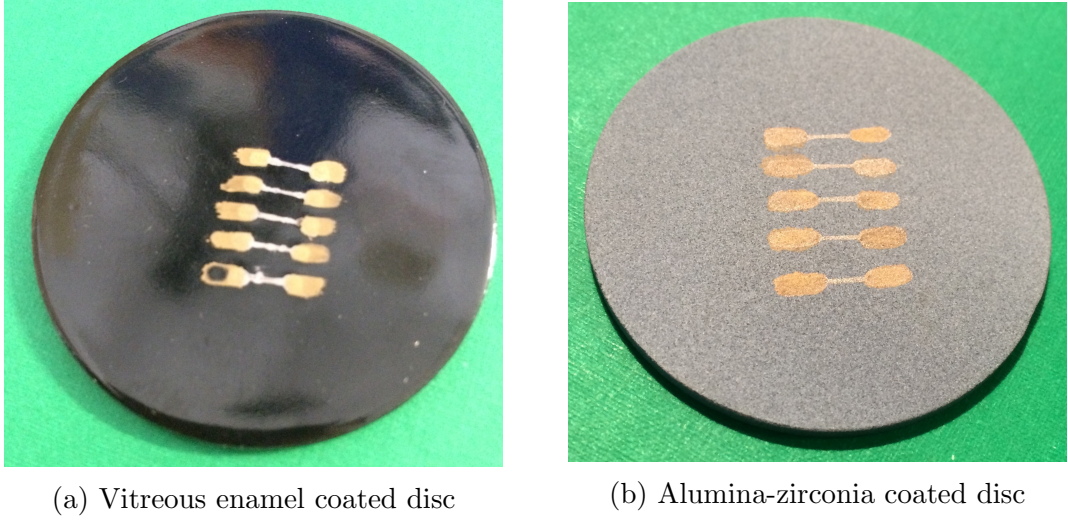


Figure 6.6: Test discs instrumented with five thin film gauges

Application of the platinum thin film gauges on the alumina-zirconia coating was a much more repeatable process, with little or no rework required after the initial application of the metallo-organic platinum and gold inks. It was observed that the surface roughness of the coating caused diffusion of the liquid platinum and gold inks which made it difficult to control the shape of the conducting track. This is evidenced by the diffuse edges of the resulting conducting tracks.

It was noted that smoothing of the surface using fine emery paper was able to reduce the approximate surface roughness of the surface from $R_a = 6.3 \mu\text{m}$ to $R_a = 1.2 \mu\text{m}$. This was observed to improve the consistency of the liquid platinum and gold ink application stage.

6.6.4 Calibration of gauges on test disc

An array of five platinum thin film gauges with gold leads were deployed on the test discs as described in Section 6.6.3. Using the method described in Section 6.4, these gauges were then calibrated to gain the temperature coefficient of resistance α_{tc} and reference resistance value R_0 (taken to be 0°C) for each gauge.

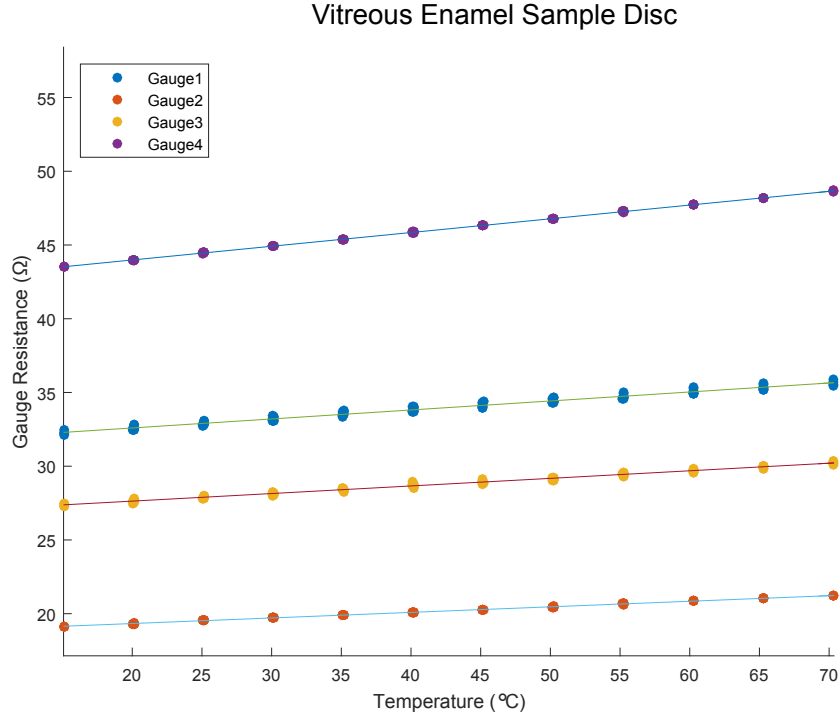


Figure 6.7: Calibration for vitreous enamel coating

Figure 6.7 shows the calibration results for the platinum gauges on the vitreous enamel coating. Each gauge required different amounts of rework before a functioning gauge was created, with one gauge deemed to have failed beyond repair. The challenges of producing a repeatable gauge is evident from the significant variation of average resistances amongst the gauges located on the same disc. The variability is caused by the deposition and firing of additional platinum at each rework stage which causes the gauge resistance to decrease.

The gauges with the lower resistance (see Figure 6.7) have undergone more rework stages. Whilst other factors will also influence the average gauge resistance (such as the width and path length of the gauge), the effect of additional platinum deposition on the gauge resistance renders these effect negligible. The values of R_0 and α_{tc} for the vitreous enamel test disc can be found in Appendix B.

Figure 6.8 shows the calibration of the platinum gauges on the raw alumina-zirconia coating (with approximate surface roughness $R_a = 6.3 \mu\text{m}$, measured using a texture comparison). It was observed that not all of the gauges displayed consistent behaviour and two of the 5 gauges (Gauge2 and Gauge3) showed considerable variation in re-

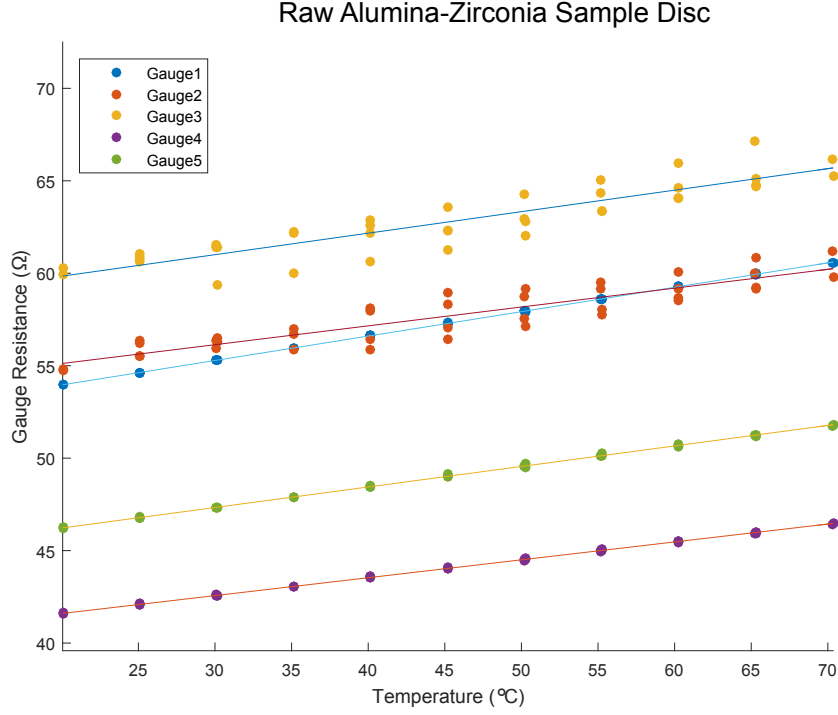


Figure 6.8: Calibration for raw alumina-zirconia coating

sistance at the same temperature, indicating problems with the consistency of the gauge. The R_0 and α_{tc} values for the gauge deposited on the raw alumina-zirconia coating can be found in Appendix B.

Figure 6.9 shows the calibration of the platinum gauges on the alumina-zirconia coating after the latter has been smoothed to approximately $R_a = 1.2 \mu\text{m}$ using emery paper. It can be seen that the variation between the gauges applied to the surface of the smoothed alumina-zirconia coating was much smaller than for both the raw alumina-zirconia and vitreous enamel coatings (see Figures 6.7 and 6.8). This is an indication that fewer stages of rework were required during the manufacturing process. The R_0 and α_{tc} values for the gauge deposited on the smoothed alumina-zirconia coating can be found in Appendix B.

It can be noted that the temperature coefficient of resistance, α_{tc} , for the thin film platinum gauges are far lower than the nominal bulk value of platinum ($\alpha_{tc} = 3.279 \times 10^{-3}$). This is due to the thin film effects as outlined by Neugebauer and Webb [97].

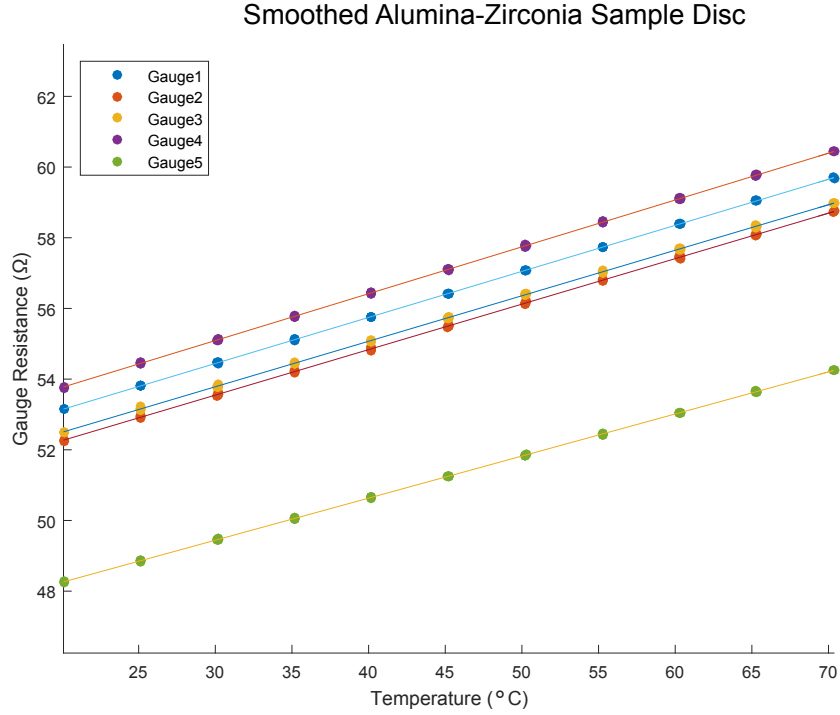


Figure 6.9: Calibration for smoothed alumina-zirconia coating

6.6.5 Coating of the probe geometry

The disc trials showed that operational platinum sensors could be manufactured on both of the investigated coatings. The next stage was to apply both the alumina-zirconia and vitreous enamel coatings to the baseline probe geometry described in Section 6.5. The successful application of the coating to the probe geometry is clearly essential for the implementation of a successful probe. The introduction of significant surface curvature which needed to be coated presented additional challenges.

Figures 6.10a and 6.10b show examples of the probe geometry instrumented with thin film gauges and coated with the vitreous enamel and the alumina-zirconia coating respectively.

6.6.5.1 Application of vitreous enamel to probe geometry

Figure 6.11 shows optical microscope images from the probe head geometry which has been coated with vitreous enamel. The probe is instrumented with two thin film gauges. The images show the various iterations of platinum application and firing



(a) Vitreous enamel coating



(b) Alumina-zirconia coating

Figure 6.10: Coated probes instrumented with thin film gauges

which causes the layering evident on both gauges.

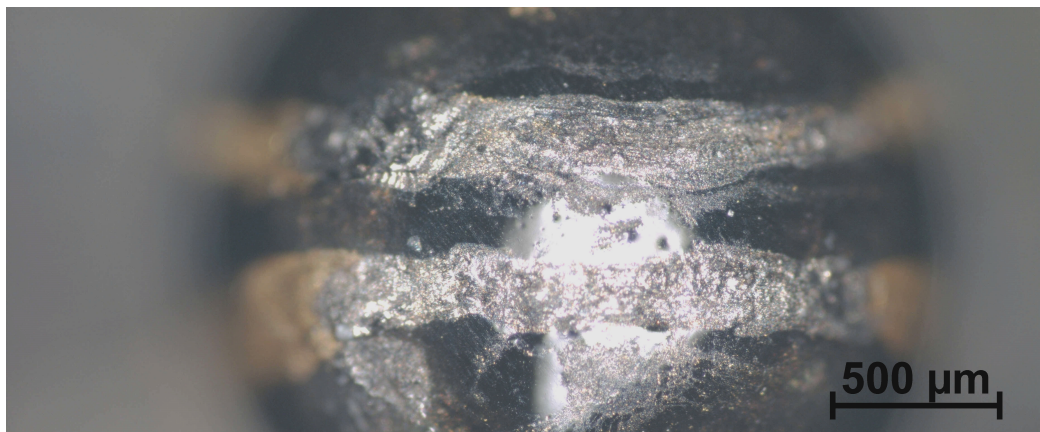


Figure 6.11: Optical microscopy of vitreous enamel coating

Figure 6.12 shows scanning electron microscope images at the center of a gauge applied to the vitreous enamel coating. It is possible to see how brittle the vitreous enamel coating is, with various cracks visible throughout the surface. However, the low surface roughness of the vitreous enamel coating is clearly evident.

The deposition of the vitreous enamel coating is difficult due to the high viscosity of the vitreous enamel at the time of application (before it is fired). This makes it difficult to accurately and consistently achieve a repeatable coating thickness and geometry due to significant curvature at the probe tip. The viscosity of the vitreous

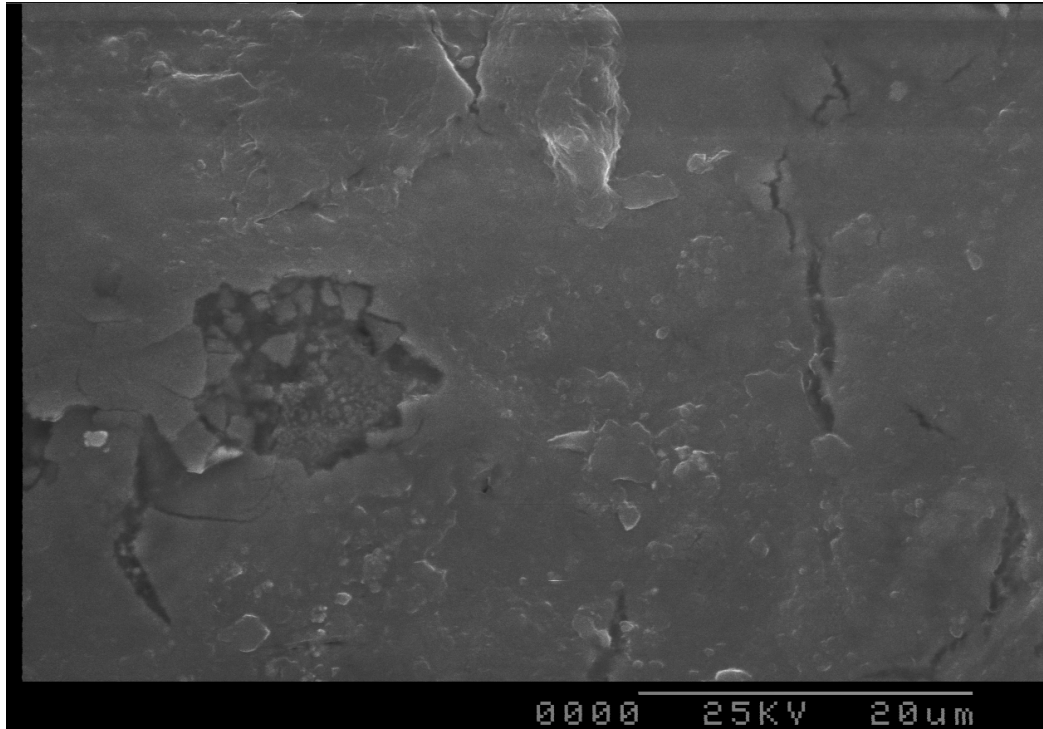


Figure 6.12: Scanning electron microscopy of vitreous enamel coating

enamel prevents the coating from accurately following the corner geometry. This resulted in the coating increasing the radius of curvature at the probe tip. It was observed that on certain examples, this results in the diameter of the probe tip being larger than specification (see Section 6.5), which interfered with the mount geometry.

An additional problem that was observed with the vitreous enamel application on the base probe geometry is caused by the gravitational effects when the liquid vitreous enamel is first applied to the geometry before the initial firing. Gravity causes the unfired vitreous enamel to run down the sides of the geometry before it solidifies in the furnace which caused variation in the coating thickness across the surface.

In addition to the coating migration before and during the initial firing, some liquefaction of the vitreous enamel surface was observed during the firing of the platinum and gold metallo-organic ink. This resulted in further failures which reduced the gauge manufacture yield as the liquefaction resulted in gaps in the conducting tracks which caused them to go open circuit. Due to the aforementioned problems, the yield of functioning sensors using the vitreous enamel coating was low (with only 2 functioning probes from 10 samples). It was decided that the yield of 20% was too

low and and that the vitreous enamel coating would not be pursued further for this application.

6.6.5.2 Application of alumina-zirconia coating to the base geometry

The alumina-zirconia coating is applied to the metallic geometry by plasma spray which avoids the majority of the problems observed during the application of the vitreous enamel coating. This method of application also allows a controllable and consistent thickness of coating to be achieved across multiple samples.

Figure 6.13 shows optical microscopy imagery of the probe head geometry which has been coated with the alumina-zirconia insulator and instrumented with a single thin film gauge. A single thin film gauge was chosen due to the difficulty in controlling the spread of platinum when deposited on the alumina-zirconia surface as the platinum tended to disperse across the surface during the firing process. This dispersion is evident in Figure 6.13 where a single main body of platinum can be seen across the middle of the gauge, but significant areas of platinum, which appears a lighter colour than the insulator, can also be observed outside the intended extent of the gauge.

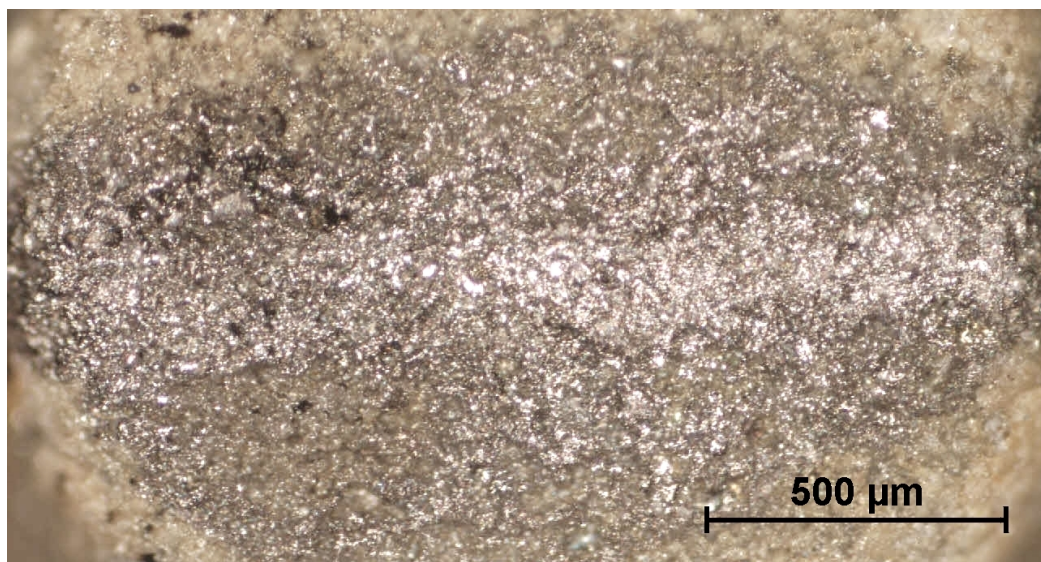


Figure 6.13: Optical microscopy of alumina-zirconia coating

The significant surface roughness of the alumina-zirconia coating can be clearly observed in Figure 6.14 which shows scanning electron microscopy at the centre of the

probe head applied with the alumina-zirconia coating. The surface profile of the coating is clearly visible with large variations in the height of the surface which can cause the deposited electrical tracks to be uneven resulting in inconsistent behaviour.

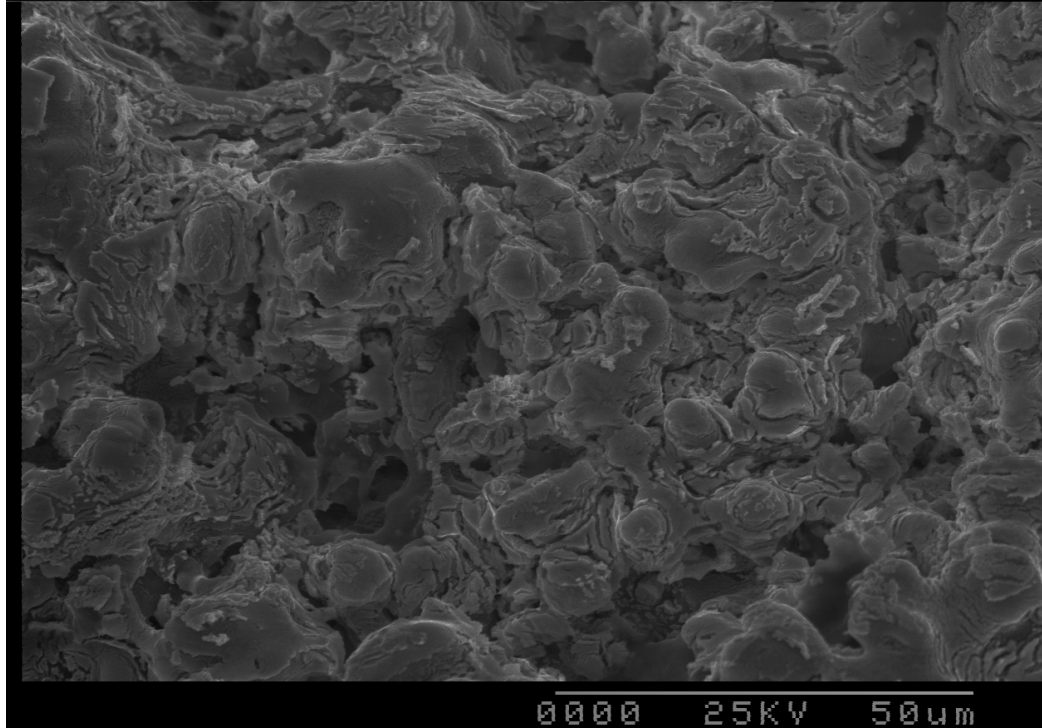


Figure 6.14: Scanning electron microscopy of alumina-zirconia coating

The surface roughness of the alumina-zirconia coating was mitigated on the test discs by manually smoothing the surface. However, to avoid problems associated with significant curvature and the edges of the probe head, the coating thickness for the probes was less than for the test discs. Reduction of the coating thickness both increases the coating adherence and reduces the intrinsic stresses, which decreases the likelihood of cracks occurring at the sharp corner. One unfortunate consequence of the reduction in coating thickness was that it then became impossible to polish the coating without breaking through to the base metal.

A common problem observed with both the alumina-zirconia and vitreous enamel coatings was the blockage of the cable routing holes during the coating process. Given the length to diameter ratio of the cable routing holes, this made it difficult to clear out the routing holes after the application of the coating as the vibrational stresses

on the drill bit. Tungsten carbide tipped drill bits were required to drill to clear any residual excess coating (due to the hardness of the coating), but several samples were scrapped due to drill bits breaking in the routing holes which could then not be retrieved.

As the alumina-zirconia coating could not be smoothed when applied to the probe geometry, the dispersion of the liquid platinum and gold metallo-organic ink was problematic during gauge manufacture. The dispersion of the gold and platinum across the surface during the firing process made it hard to create well defined conducting tracks. Cracking was also observed at the probe tip after the firing process causing the majority of the gauges deposited to be open circuit. These aforementioned problems resulted in none of the 10 samples being functional. With this mind, it was decided that the alumina-zirconia coating would not be pursued for the probe geometry.

6.7 Design and Manufacture of a Single Layer Heat Flux Sensor

6.7.1 Determination of base material

As explained in Section 6.3, both single layer and multi-layer thin film gauge systems were considered for the application to the internal combustion engine. The key advantage of using single layer systems is the simplicity associated with the characterisation of the sensor thermal behaviour. In order to obtain accurate heat flux measurements using a multi-layer system, it is necessary to accurately establish the coating thickness and the thermal properties of both materials. Though a specification is often given for the coating, any deviation from this can cause significant errors when calculating the surface heat flux.

In order to create a sensor to meet the specification outlined in Section 6.1, the use of ceramic material was considered. Ceramics offer very high mechanical strength, low porosity, smooth surface finish and good electrical insulation properties which make them ideal for thin film gauge applications. However, ceramics are typically difficult to machine owing to their high mechanical strength and abrasive resistant nature.

In the present investigation, two ceramic materials were considered: alumina (Al_2O_3) and fused quartz (SiO_2). In order to avoid the difficulty of machining the ceramic materials to fit the desired profile and instrumentation mount, a stainless steel collet (see Section 6.5) was manufactured using direct metal laser sintering (DMLS). A ceramic rod was then interference fit into the collet to fulfil the cable routing and geometry requirements allowing the sensor to conform to the specifications outlined in Section 6.1.

6.7.2 Initial observations on platinum and gold deployment

As before, the platinum and gold conducting tracks were applied using the hand painted metallo-organic ink (as described in Section 6.3). Other researchers have previously used sputtering to deposit the platinum on ceramic substrates [98], but this was deemed to be a more complex method with little additional benefit.

Two samples were used to assess the suitability of the chosen platinum and gold deployment method on the two base materials. Platinum thin film gauges and gold conduction tracks were applied to the top face of 10 mm diameter samples of length 30 mm. Chamfers were added to the edges of the top face in order to aid the gold tracks to reduce the surface curvature.

Platinum and gold application was straightforward, owing to the low porosity and low surface roughness of the base materials, with minimal surface preparation required. Each sample underwent a succession of painting and firing iterations in order to gradually increase the thickness of the platinum thin film gauge to reduce the resistance to the required specification. Both the alumina and fused quartz required three iterations (painting followed by firing) before the gauge reached an acceptable resistance (less than $100\ \Omega$). The number of iterations was found to be consistent for all the sensors manufactured and no further rework was then required as none of the gauges deposited exhibited open circuit behaviour after firing.

One difficulty was the fragility of the silver solder pad. The adherence of the solder pad to the ceramic surface was observed to be susceptible to failure under any stress. This was particularly the case for the fused quartz ceramic sample.

6.7.3 Optical and scanning electron microscopy of samples

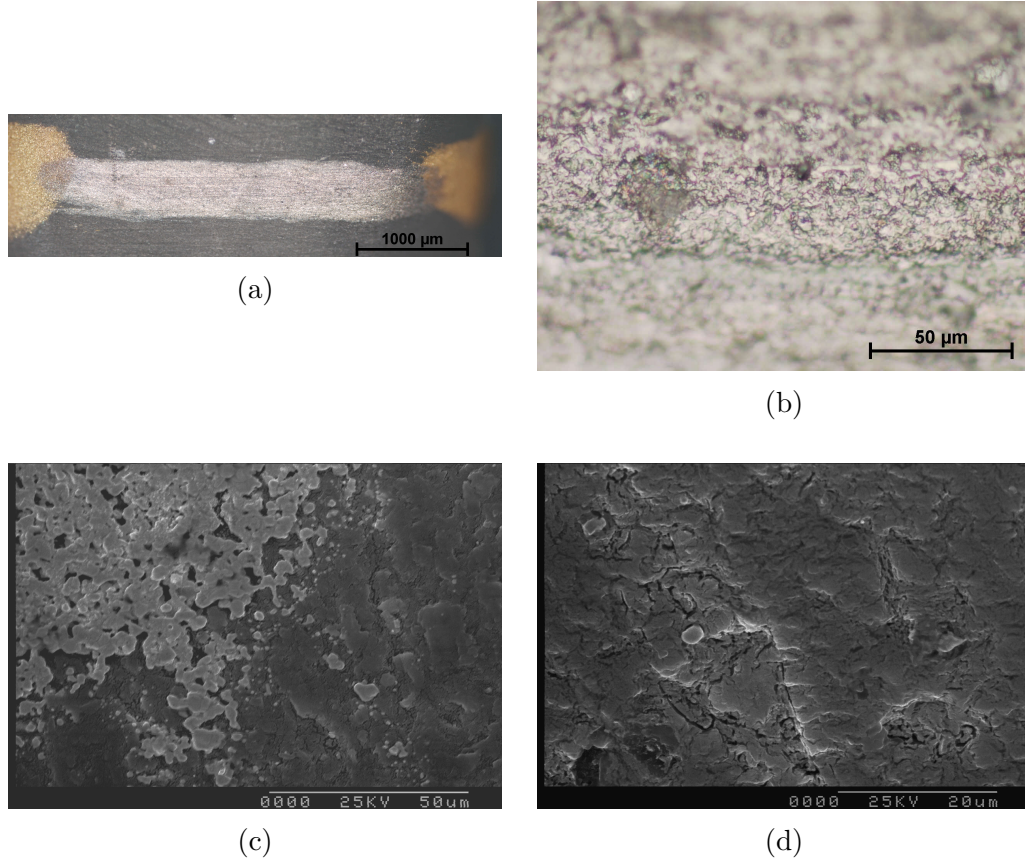


Figure 6.15: Optical and scanning electron microscopy of fused quartz sample

The fused quartz and alumina samples were imaged using both an optical and scanning electron microscope. Figure 6.15 shows a selection of images from the fused quartz sample. Figures 6.15a and 6.15b show the platinum gauge and gold conducting tracks under optical magnification. It is possible to observe the smooth surface roughness of the quartz through the layer of deposited platinum (see Figure 6.15b).

Figures 6.15c and 6.15d show scanning electron microscopy imagery of the platinum-gold interface and platinum gauge respectively. As before, it is possible to observe the smooth surface finish of the fused quartz surface with minimal variation in height evident.

Figure 6.15c shows the deposition of the gold conducting tracks on top of the platinum. It is possible to see the difference in structure after firing with the platinum spreading

to take on the profile of the surface on the fused quartz, whereas the gold tends to remain as isolated flakes.

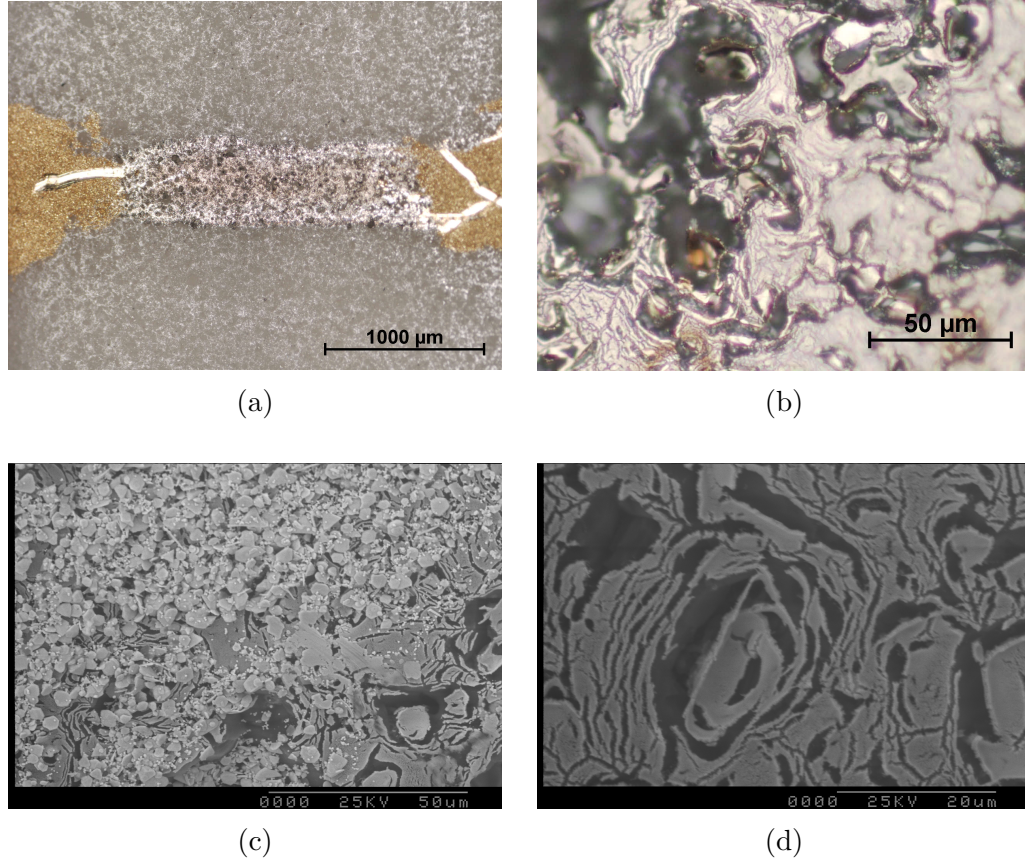


Figure 6.16: Optical and scanning electron microscopy of alumina sample

Figure 6.16 shows microscopy images of the alumina sample. Figures 6.16a and 6.16b show the platinum gauge and gold conducting tracks under optical magnification. When contrasted with the fused quartz sample (see Figure 6.15b), increased variation in surface height can be observed in Figure 6.16b.

Figures 6.16c and 6.16d shows scanning electron microscopy imagery of the platinum-gold interface and the platinum gauge as deposited on the alumina sample respectively. As before, the deposition of the gold on top of the platinum gauge is clearly evident in Figure 6.16c with the gold retaining a flake-like structure. As before, the platinum can be seen to take on the surface profile of the area in which it is deposited whereas the gold tends to retain its flake-like structure on top of the surface.

6.7.4 Calibration of thin film gauges on ceramic substrate

Figure 6.17 shows the calibration results of a platinum thin film gauge deposited on the fused quartz and alumina samples respectively. Due to the low porosity and low surface roughness of both the substrates, no issues were expected with the calibration of the gauges on the substrates. Previous work by Wrbanek and Laster [98] also indicated that the platinum deposition process on both the fused quartz and alumina has been shown to be repeatable and consistent.

Both the thin film gauges underwent an annealing process (as described in Section 6.8) prior to calibration. The 95% confidence intervals of both the temperature coefficient of resistance and the reference resistance value are small compared to the mean values calculated using linear regression indicating that the gauges are consistent across the range of temperatures.

As before, it can be noted that the temperature coefficient of resistance for the platinum gauges are much lower than the bulk value due to thin film effects. A benefit of the gauge deposition process being repeatable and predictable is that the resistance of each gauge can be kept relatively consistent (given that subsequent rework tends to decrease the resistance due to the additional platinum deposited).

The values for the calibration of the ceramic substrate test samples can be found in Appendix B.

6.7.5 Use of ceramics in heat flux sensor geometry

As described in Section 6.5, a interference fit system was used to fit a ceramic core into a stainless steel collet. An example of the core (made from alumina and instrumented with a single thin film gauge) is shown in Figure 6.18a. The ceramic cores were first instrumented with a thin film gauge and calibrated before being assembled with the collet.

In order to fit the alumina core, the stainless steel collet was first heated in order to allow the alumina core to be inserted. The orientation and position of the alumina core was such that the solder joins for the thin film gauge were between the castellations on the collet, allowing the connecting wires to be threaded through the routing holes.

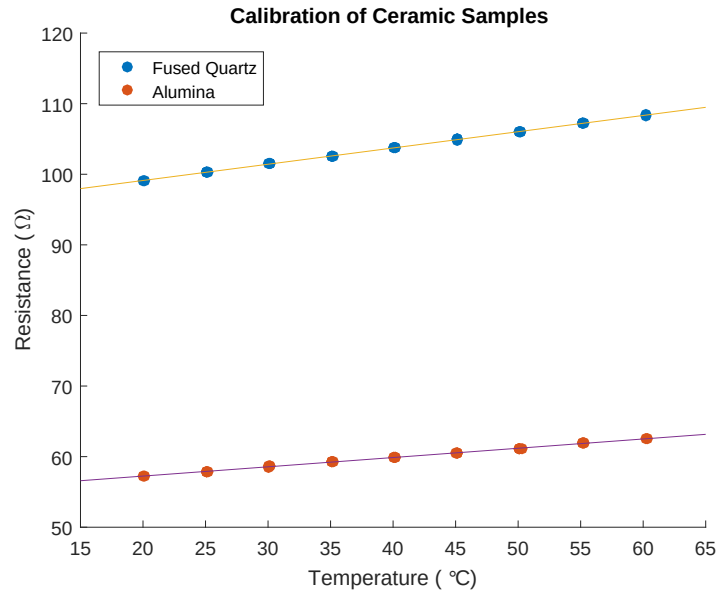


Figure 6.17: Calibration of thin film gauges on ceramic samples

It is imperative that the solder joint does not touch the collet as this would cause the gauge to short circuit with the base metal. The collet was then allowed to contract around the core in order to secure it in place.

Ceramic adhesive was placed on the bottom of the collet to further secure the ceramic core in place and to secure the connecting wires in place (reducing the risk of fatigue failure of the solder joint). Additional ceramic adhesive was placed over each solder joint between the castellations to further protect the solder joints. This adhesive also helps to keep the ceramic gasket in place. A complete assembled example can be seen in Figure 6.18b.

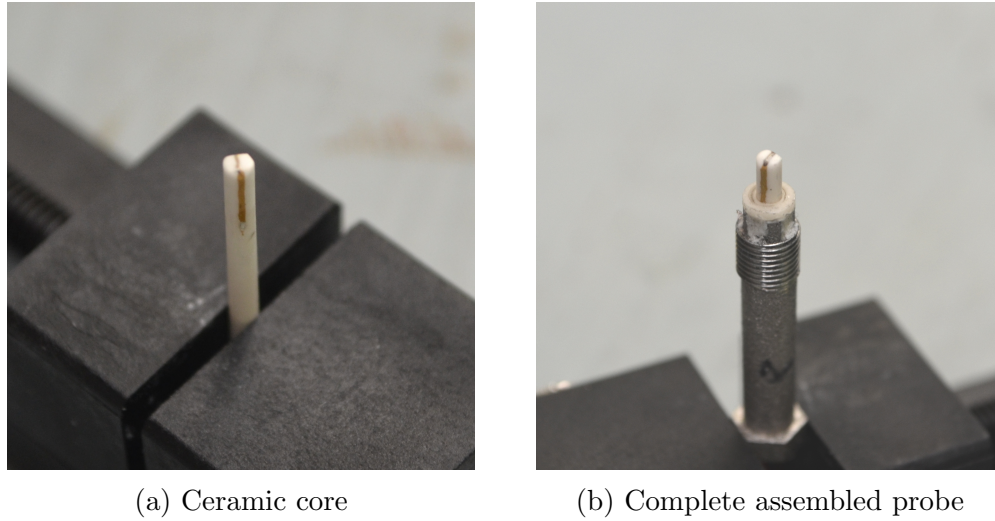


Figure 6.18: Alumina ceramic sensor

6.8 Effect of Annealing on Platinum Thin Film Gauges

The effect of annealing the platinum thin film gauges was investigated. This was performed by subjecting the platinum thin film gauges to an elevated temperature for an extended period of time in the thermostatic water bath before the calibration phase. The annealing phase consisted of submerging the platinum thin film gauges in the thermostatic water bath at 80°C for 8 hours prior to calibration.

Table 6.2: Annealed thin film gauges on smoothed alumina-zirconia coating

Gauge Number	Non-Annealed		Annealed	
	α_{tc} 95% confidence value	R_0 95% confidence value (Ω)	α_{tc} 95% confidence value	R_0 95% confidence value (Ω)
1	0.000030	0.073	0.000002	0.005
2	0.000023	0.054	0.000012	0.028
3	0.000031	0.074	0.000009	0.022
4	0.000031	0.076	0.000008	0.019
5	0.000032	0.071	0.000005	0.010

Annealing has been shown to improve the electrical characteristics of platinum thin

film gauges by relaxing intrinsic stresses within the metal thin film, caused by mis-aligned grains or islands incorporated at the low temperature liquid deposition stage. Realignment of the material micro structure through annealing has been shown to reduce the variation in the performance of the thin film gauge [99].

Although the process described above can be considered to be much milder than the annealing processes typically described in literature [99] [100], the results in Table 6.2 show that the annealing cycle still yields significant improvements. The 95% confidence values for the thin film gauges post-annealing show significant reductions in both the temperature coefficient of resistance, α_{tc} , and reference temperature, R_0 . This shows an increase in the consistency of the gauge at each temperature step during the calibration.

Chapter 7

Thin Film Gauge Experimental Data

7.1 Response Time Testing of Example Thin Film Gauges

As described in Chapter 6, the proposed heat flux sensor was specified to have a response frequency of 100 kHz in order to provide crank angle resolved data at 15,000 RPM.

In order to characterise the response time of the sensor, the impulse response of an example thin film gauge was tested using a pulsed signal from a high power LED. A similar arrangement is described by Buttsworth et al. to test the impulse response of eroding thermocouples [49]. An overview of the test is shown in Figure 7.1.

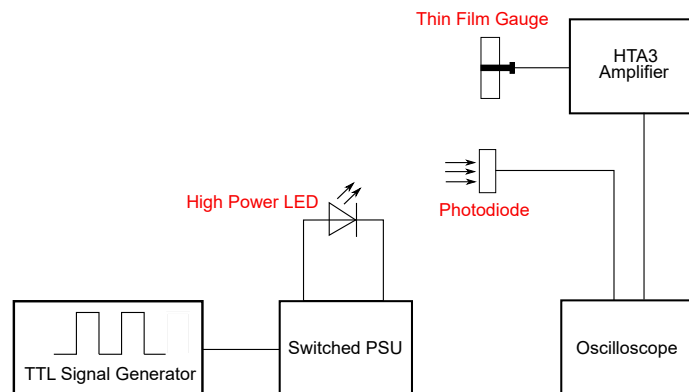


Figure 7.1: Overview of impulse response test facility

The thin film gauge was located 50 mm perpendicularly from the high power LED. This was connected to a low noise, differential amplifier in order to boost the output signal from the thin film gauge [101]. This was necessary due to the low sensitivity of the thin film gauge at high frequencies. The original signal can then be recovered by using the impulse response method [95].

A photodiode with a 10 ns response time was located the same distance from the high power LED as the thin film. The function of the photodiode was to enable the phase of the LED heat flux to be determined.

The high power LED was then triggered by a TTL signal generator in order to provide a step change in radiant heat flux on the thin film gauge. The incident beam from the high power LED is also received by the photodiode, which responds immediately. The resulting data was logged using a 8 bit, 30 MHz digital oscilloscope¹.

The resulting temperature data was then post-processed using the impulse response method (described in Section 6.3.1) in order to calculate the heat flux.

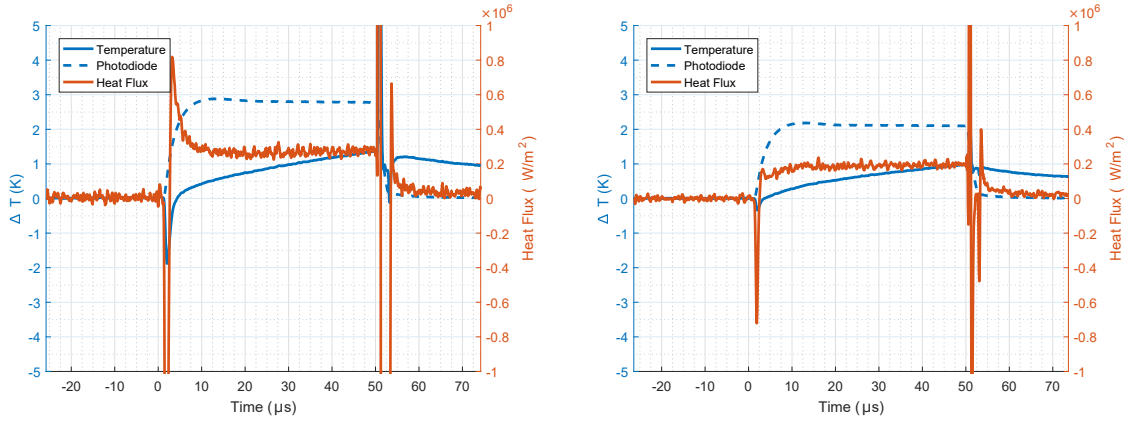


Figure 7.2: Comparison between thin film gauge and photodiode response. It should be noted that the photodiode response has been scaled and has arbitrary units

Figure 7.2 shows a comparison between the photodiode and thin film gauge response for a step change in radiant heat flux for two thin film gauges.

The resistance of the thin film gauges (at 20°C) was measured to be 17.4 Ω and 19.8 Ω . Low resistances are not desirable as the resistance of the gauge is inversely

¹Tektronix TBS1032B digital oscilloscope

proportional to the thickness of the deposited platinum. Thicker layers of platinum result in slower response times due to the increase in the thermal mass. Therefore, it was assumed that any thin film gauges with a higher resistance would perform better than the thin film gauges tested here.

The thin film gauge must be sufficiently thick to satisfy the requirement for the resistance to be lower than $100\ \Omega$. This is due to practical constraints on the voltage output from the HTA3 amplifier [101]. As the power supply has a relatively limited voltage range, a higher resistance reduces the current that can be supplied to the thin film gauge. This, in turn, reduces the resulting signal strength from the thin film gauge.

Various features of the signal can be seen at the rise and fall of the step change due to noise from the switching of the power supply. As the photodiode responds almost instantaneously to the incident beam from the high power LED, the response of the thin film gauge can be compared directly to this signal. It can be seen that the response of the thin film gauge lags behind the photodiode which is to be expected. This lag was observed to be consistently between $5\text{--}10\ \mu\text{s}$ which was considered to be acceptable to meet the specified response frequency of $100\ \text{kHz}$. This matches with similar experiments by Penty-Garaets et al. [102].

7.2 Engine Testing

The thin film gauges were tested on a small single cylinder test bench. The test bench comprised a Honda GX160 air-cooled, 4-stroke engine powering a $2\ \text{kW}$ electrical generator unit. A $2\ \text{kW}$ electrical heater was connected to the generator in order to provide a resistive load.

Table 7.1 shows the specifications of the GX160 single cylinder test bench. The cylinder head was modified to include a tapped port for a piezoelectric pressure transducer² (as described in Section 6.5), which matches the shape of the heat flux sensor described in Chapter 6. The mounting for the instrumentation was aligned parallel with the valve stems in the cylinder head and perpendicular with the piston direction

²Kistler 6054BR

Table 7.1: GX160 engine specification from the manufacturer

Type	1-cylinder, 4-stroke
Combustion system	Spark-ignition, carburetor
Bore $\times$ stroke	68 $\times$ 45 mm
Connecting-rod length	86 mm
Compression ratio	9:1
Fuel	Gasoline

of motion (see Figure 7.3b). The instrumentation mountings were designed to allow readings in the clearance volume of the cylinder.

Low noise differential amplifiers were used in order to amplify the output signal from the thin film gauges, as in Section 7.1. The probes used to gather the data were pure alumina gauges (as described in Section 6.7). The calibration of the probes used can be viewed in Appendix B.

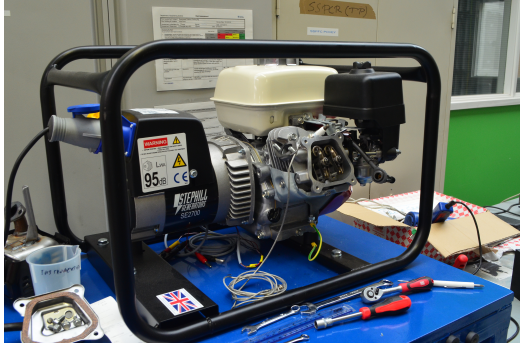
The test bench was additionally fitted with an inductive NPN junction sensor to allow location of top dead center during engine running. The induction sensor is triggered by a piece of aluminium located on one of the plastic flywheel blades. The data from the induction sensor is then interpolated to generate the crankshaft location throughout the engine cycle (assuming constant rotational speed between each detection of the the TDC location). This then allows calculation of the cylinder volumetric data throughout the cycle.

The data was recorded using a 4 channel, 24-bit data logger³ which allows synchronous recording of the 4 channels simultaneously at 100 kHz.

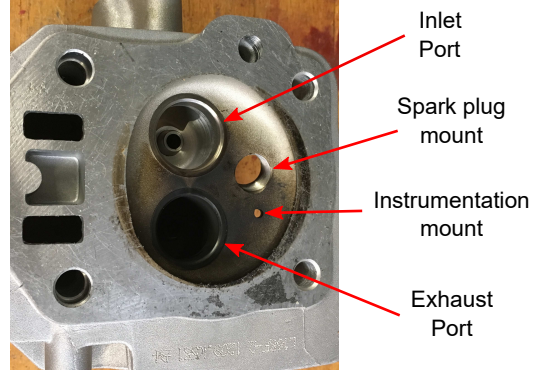
7.2.1 Comparison with empirical heat transfer models

There have been numerous investigations into the heat transfer mechanisms that occur inside the cylinder of an internal combustion engine. These investigations have yielded numerous heat transfer correlations that are used to predict the heat fluxes that arise during the combustion process. A typical approach is to correlate heat flux to known engine parameters using a power law with empirically determined

³NI CompactDAQ 9174



(a) GX160 single cylinder test bench with 3 kW generator unit



(b) Instrumentation mounts in the cylinder head (indicated by arrow)

Figure 7.3: GX160 single cylinder test bench

coefficients and exponents. The heat flux data recorded from the single cylinder test bench was compared against two of the most widely used empirical correlations.

The Woschni heat transfer model [5] was developed to predict the heat fluxes that arise during the combustion phase of the engine cycle. The heat transfer coefficient is predicted using the in-cylinder pressure, temperature, reference gas velocity, and characteristic length of the cylinder using the expression in Equation 7.1.

$$h = C_0 p^{0.8} u^{0.8} B^{-0.2} T_g^{-0.53} \quad (7.1)$$

The reference velocity, u , throughout the engine cycle is calculated using the expression shown in Equation 7.2. The value of coefficients C_1 and C_2 in Equation 7.2 are dependent on the phase of the engine cycle (see Table 2.1).

$$u = C_1 v_{p,m} + C_2 \frac{V T_r}{p_r V_r} (p - p_m) \quad (7.2)$$

The Hohenberg heat transfer model [6] is another widely used empirical heat transfer model used to predict the heat fluxes during the engine cycle. The Hohenberg model simplified the model proposed by Woschni by removing the dependence on the instantaneous reference gas velocity (see Equation 7.2). In addition, the characteristic length used was changed from the cylinder bore (used by Woschni) to the diameter

of a sphere with the same instantaneous volume as the cylinder. Heat transfer coefficients predicted by the Hohenberg model are calculated using the expression shown in Equation 7.3.

$$h = C_0 \frac{p^{0.8}}{V^{0.06} T^{0.4}} (v_p + 1.4 \text{ m/s})^{0.8} \quad (7.3)$$

In both cases, the C_0 term is a scaling factor used to tune the empirical model. In order to calculate the C_0 term, the heat flux was measured on the single cylinder test bench in the motored condition. A C_0 term was then calculated for each heat transfer model in order to match the predicted peak heat flux with the measured peak heat flux.

For both empirical models, the in-cylinder pressure, cylinder volume, cylinder geometry, and mean piston velocity are all known and the cylinder gas temperature can be calculated using the ideal gas law. Both the Woschni [5] and Hohenberg [6] models are single-zone models which assumes the gas temperature is homogeneous throughout the cylinder. The cylinder wall temperature, T_w , was assumed to be 300 K for cold start and motored conditions, and 400 K for the firing condition. Using this aforementioned variables, the heat transfer coefficients predicted using the Woschni and Hohenberg models can then be used to calculate the heat flux into the cylinder wall throughout the engine cycle using Equation 7.4.

$$q = h (T_g - T_w) \quad (7.4)$$

Figure 7.4 shows the measured heat flux during the motored condition. This was then used to calculate the C_0 coefficients for both the Woschni and Hohenberg heat transfer models. These were found to be 13.7 and 5.0 respectively. These scaling factors were then used to predict the heat flux that would occur under firing conditions. It should be noted that the motoring condition was induced by use of the recoil starter and, as such, was limited to 700 RPM which is lower than the operating speeds of the engine under firing conditions.

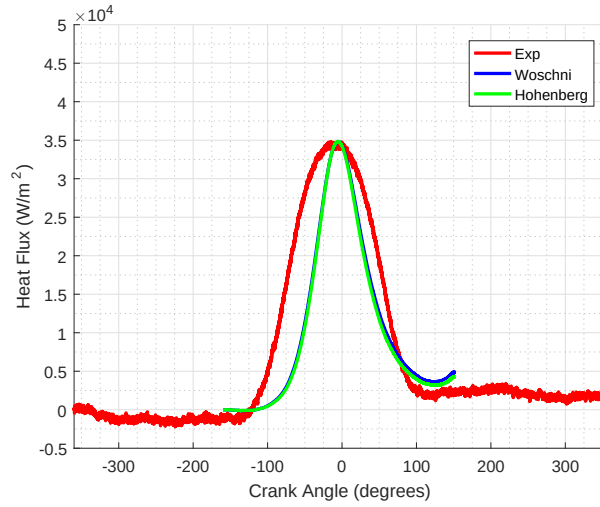


Figure 7.4: Heat flux comparison for motored condition

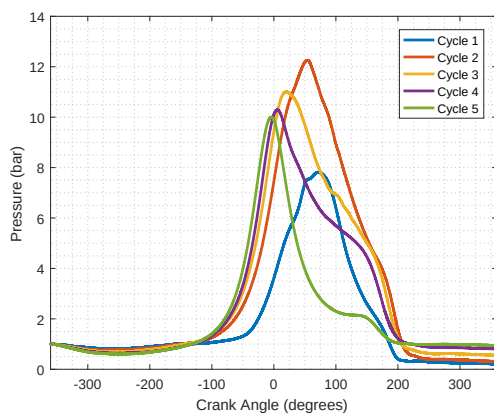
Figure 7.4 shows that the peak magnitude of the measured heat flux is well-matched with the theoretical models, which is expected given the motored condition is used to tune the theoretical models. However, it can be seen that the profile of the theoretical heat flux profile also matches well with the measured data.

7.2.2 Cold start conditions

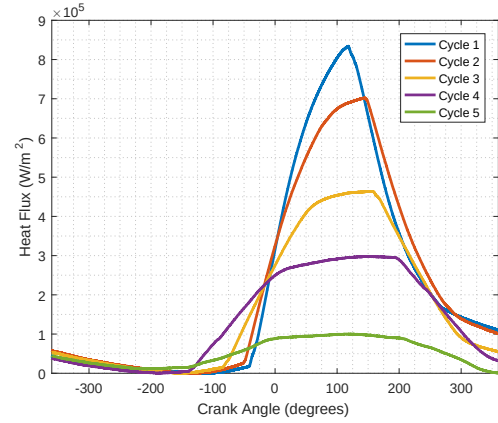
Figures 7.5a and 7.5b show the measure in-cylinder pressure and heat flux respectively for the first 5 cycles after the engine start, running at 3000 RPM. The heat flux can be seen to decrease throughout the first 5 cycles as the pressure in the inlet manifold begins to decrease. The decrease in inlet manifold pressure is due to the closed choke (required to increase the richness of the air-fuel mixture) in the cold start condition. This increases the likelihood of incomplete combustion, resulting in reduced in-cylinder heat flux and pressure.

Figure 7.5d shows the heat flux for a single cycle for the engine in the cold start condition. It can be seen that the heat flux predicted using the Woschni and Hohenberg models match up well with the measured heat flux data. It can be seen that the rise of the measured data occurs before that of the empirical models. This is likely to be caused by the location of the thin film gauge in the cylinder.

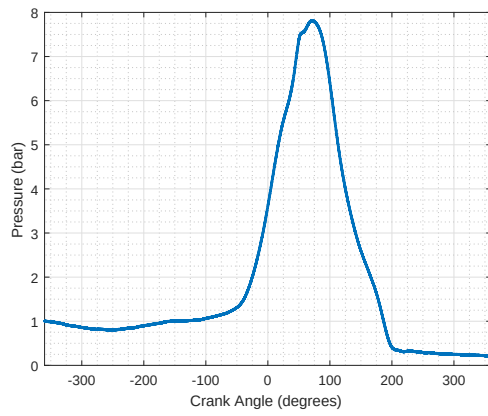
The single-zone models assume a spatially averaged heat transfer coefficient in the



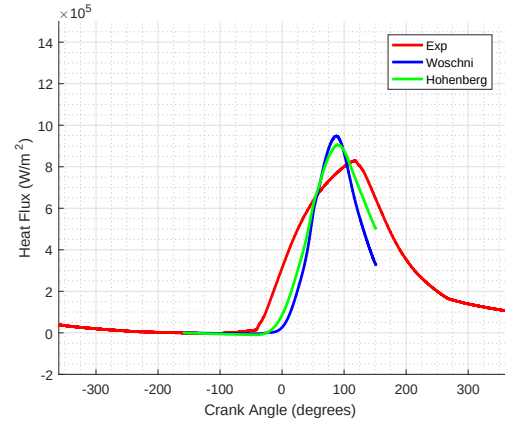
(a) Measured in-cylinder pressure for the first 5 cycles during cold start condition



(b) Measured heat flux for first 5 cycles during cold start condition



(c) Measured in-cylinder pressure during cold start condition



(d) Heat flux comparison for cold start condition

Figure 7.5: Measured single cylinder data for cold start condition

cylinder, whereas there is likely to be spatial variation due to the propagation of the flame front during the combustion phase. As the thin film gauge is located very close to the spark plug (the location from which the flame will propagate), the measured heat flux is likely to rise quicker than the spatial average.

7.2.3 Firing condition

Figure 7.6a shows the measured in-cylinder pressure for multiple cycles during fired engine operation at 3000 RPM, with good repeatability throughout continuous cycles. Figure 7.6b shows the average measured heat flux during engine operation at 3000 RPM over multiple cycles. The measured data is compared to the empirical-based

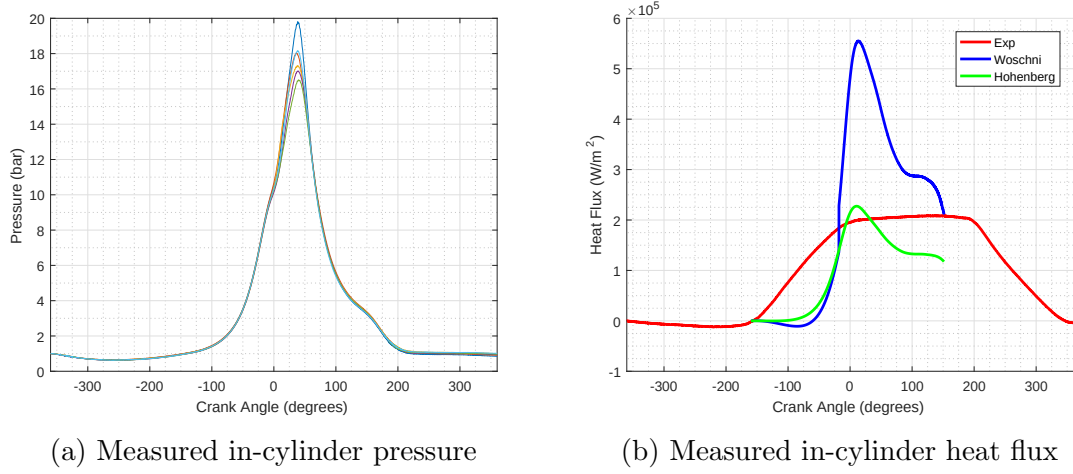


Figure 7.6: Measured single cylinder data for firing condition at 3000 RPM

Woschni and Hohenberg heat transfer models as before. It can be seen that there is a large discrepancy between the Woschni model and the measured data, whilst the Hohenberg model predicts the peak magnitude of the measured heat flux within 10%. This follows the trend reported by Hohenberg [6], who found that the Woschni correlation tended to under estimate the heat flux during the compression and expansion phases of the engine cycle, whilst over predicting the heat flux that occurs during the combustion phase. An investigation by Schmitt et al. [103] also concluded that the Hohenberg model has consistently tended to show better agreement than the Woschni model for experimental results from modern engines.

The heat flux can be seen to rise throughout throughout the compression phase before reaching a plateau throughout the combustion and expansion phases. The heat flux then begins to fall during the exhaust phases and intake phases as the cylinder is filled with air at ambient temperature. The plateau throughout the combustion phase does not agree with either theoretical heat transfer model, which both indicate a sharp peak followed by a decrease in heat flux during the expansion phase. A possible contributing factor to this behaviour could be the gradual erosion of the platinum thin film gauge during engine operation which would cause the gauge resistance to increase, leading to erroneously high temperature measurements.

Figure 7.7 shows published heat flux measurements from the cylinder head of a four-cylinder spark ignition engine operating at 3000 RPM using thin film gauge sensors

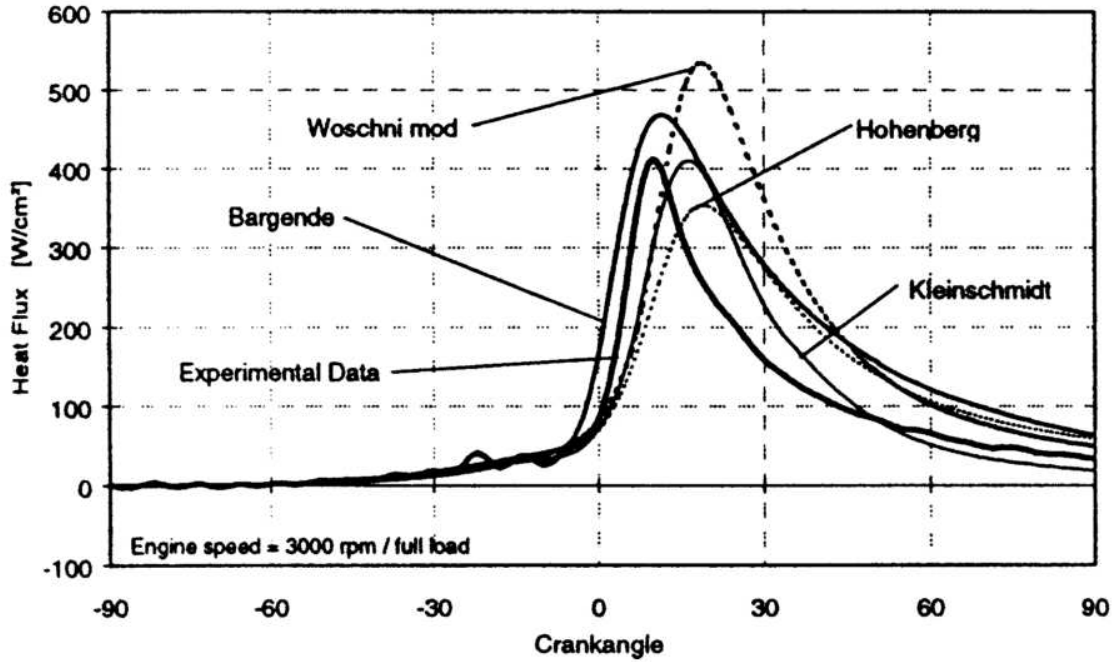


Figure 7.7: Measured cylinder head heat transfer in a four-cylinder spark ignition engine operating at 3000 RPM [34]

[34]. As in this study, the measured values are compared to heat transfer predictions calculated using empirical heat transfer models.

It can be seen that the profile of the experimental data is similar to the values predicted by the empirical heat transfer models, unlike the data shown in Section 7.2.3. This indicates that the different heat flux signal profile shown in Figure 7.6b may have been caused by issues with sensor robustness during engine operation. Similar to the results shown in Section 7.2.3, it can be seen that the Woschni model [5] overpredicts the peak heat flux observed during the engine cycle, while the Hohenberg model [6] provides a closer estimation of the maximum heat flux value.

7.3 Thin Film Gauge Robustness

As described in Chapter 6, the measurement of heat fluxes inside an engine cylinder presents a variety of different challenges due to the aggressive environment.

Each experimental run was 30 seconds in duration. Each sensor was subjected to multiple experimental runs until the thin film gauge became open circuit, at which

point the sensor was swapped out for a different sensor. After each sensor was removed from the single cylinder rig, it was observed that areas of the thin film gauge had been removed from the surface of the alumina substrate, causing the sensor to become open circuit. It was thought that there were two possible reasons for these failures.

The first possible cause is that the temperature of the sensor is increasing to the point where the platinum is de-laminating from the ceramic substrate. This is caused by the low thermal conductivity of the alumina substrate, which allows the temperature of the surface exposed to the combustion gases to rapidly rise in temperature. A low thermal conductivity was necessary for this set of experiments as it allows the transient temperature change to be modelled as 1-D for a longer duration.

Work by Gafaar looked at the high temperature performance of platinum resistive elements on alumina substrates [104]. It was concluded that the platinum was likely to de-laminate from the alumina substrate at a temperature of approximately 800 °C.

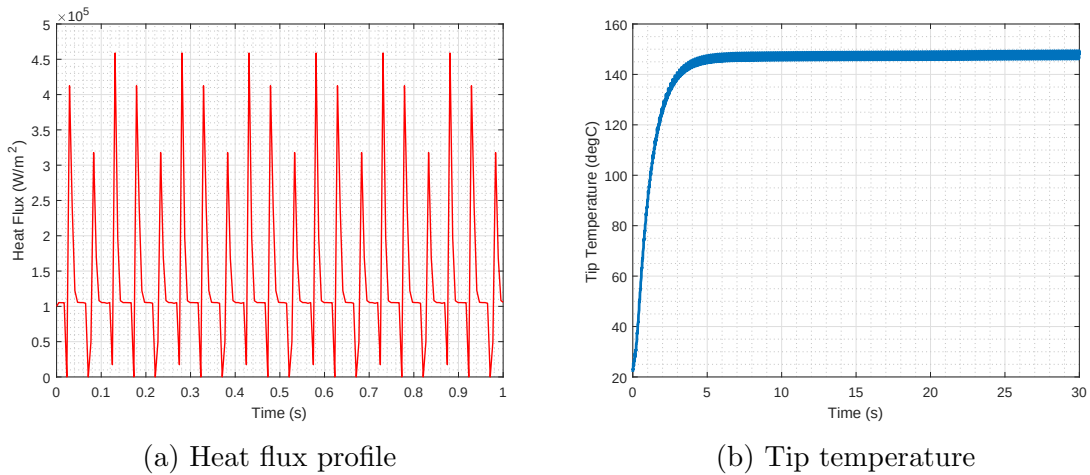


Figure 7.8: 3-D FEA simulation of alumina tip temperature

In order to predict the temperature of the ceramic during the experiment, a 3-D finite element analysis (FEA) model was created. Figure 7.8b shows the tip temperature of the alumina rod (at the gauge location) when subjected to a representative engine cycle heat flux profile with a peak heat flux of 450 kW/m² (shown in Figure 7.8a). The heat flux profile was generated using a Simulink model of an internal combustion engine that was adapted from work by Buttsworth et al. [105].

It can be seen that the temperature is not expected to exceed 150 °C which is far below the expected de-lamination temperature of the platinum thin film gauge. Therefore, it was concluded that the temperature of the thin film gauge was not a limiting factor of the gauge robustness. The second possible explanation was that the platinum was being gradually removed from the ceramic substrate by abrasion during the engine cycle.

7.4 Application of an Amorphous Diamond Coating

7.4.1 Introduction to diamond shielded thin film gauges

In order to increase the robustness of the gauge to abrasion during the engine cycle, a novel amorphous diamond coating was applied over the top of the thin film gauge. Use of a chemical vapour deposition (CVD) diamond to shield thin film gauges from abrasive environments in hypersonic test facilities was first investigated by Clark [106] and more recently by Penty Geraets et al. [102].

The use of diamond to shield thin film gauges from direct contact with the test gas is possible due to its high thermal conductivity, which is approximately 2000 W/mK at room temperature [107]. Diamond is also electrically insulating and mechanically hard allowing the thin film gauge to be applied directly to the diamond, with the test gas in contact with the reverse side. The thickness of the diamond is the key factor in determining the response time of the sensor, with thicker diamond films having a slower response time. The consequence of having a thinner diamond film is that the resulting sensor tends to be more delicate. Work by Penty Geraets et al. has tested the use of CVD diamond films from 75 µm to 300 µm thickness [102].

7.4.2 Amorphous diamond application method

The disadvantage of the approach outlined by Penty Geraets et al. is the cost and delicacy of the CVD diamond. As such, a CVD applied amorphous diamond coating (supplied by Diamond Hard Surfaces Ltd.) was trialled. The use of an amorphous diamond coating (as opposed to a pure diamond) was adjudged to have significant

advantages in terms of solution robustness. An example of a ceramic gauge, coated with amorphous diamond, is shown in Figure 7.9.

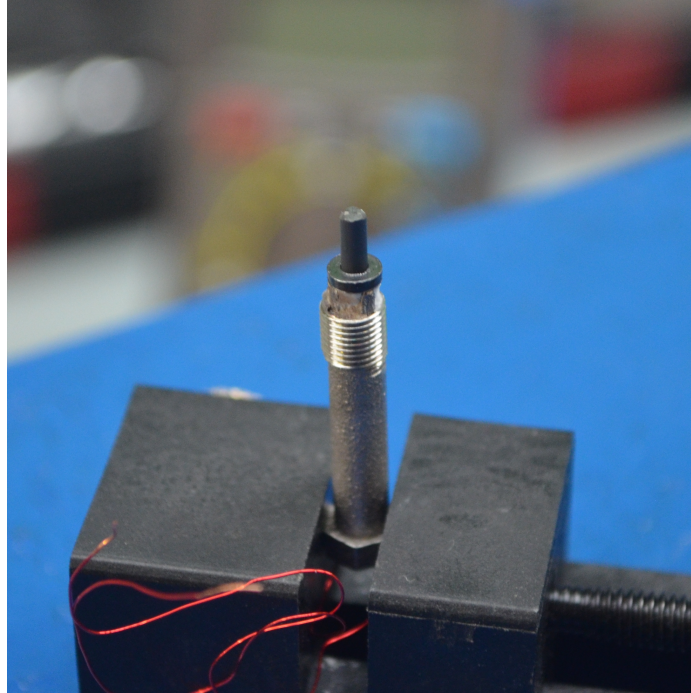


Figure 7.9: Complete amorphous diamond coated ceramic probe

The use of CVD allows the coating to be applied at relatively low process temperature (approximately 100°C). As the coating is applied after the platinum and gold tracks have been deployed, it was important that the application of the amorphous diamond coating did not impact of the quality of the sensing circuit.

Due to the smoothness of the ceramic substrate, minimal surface pre-processing was required in order to get the diamond coating to adhere to the ceramic substrate. An ion-bombardment sputtering process was applied for two minutes, in order to clean the surface before the amorphous diamond coating was applied. This had the effect of removing some of the conducting circuit (evidenced by an increase in resistance of the thin film gauge after the diamond coating had been applied). However, if the sensing circuit was intact and the resistance remained below $100\ \Omega$, this effect was not considered to be problematic if the coated sensor passed calibration.

The calibration of the diamond gauge is shown in Appendix B. It can be seen that the confidence values of the thin film gauge are higher than observed for the pure

ceramic gauges. This uncertainty is thought to be caused by the sputtering process used to prepare the surface for coating. As this procedure erodes the surface, this can lead to inconsistencies in the sensing circuit caused by the path of the beam.

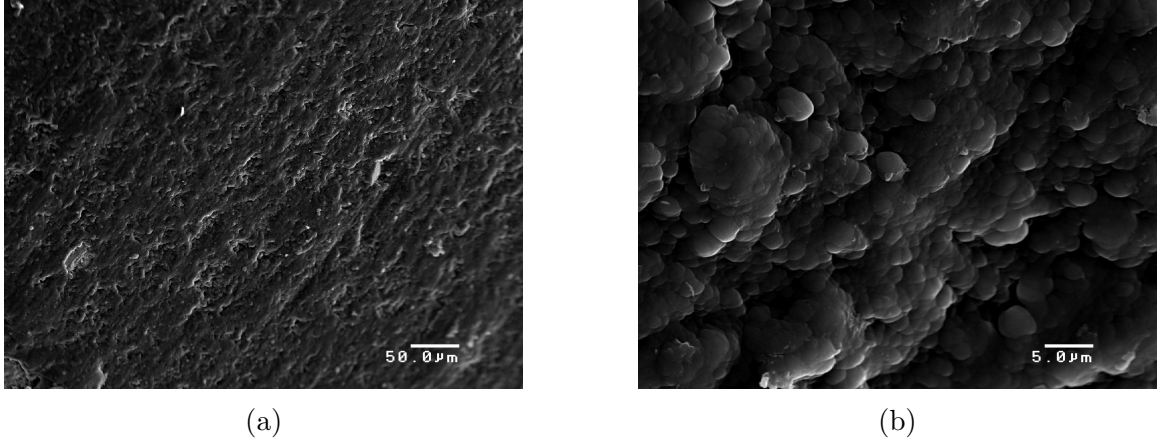


Figure 7.10: Scanning electron microscopy of amorphous diamond coating

Figure 7.10 shows scanning electron microscopy of the amorphous diamond coating. The smoothness of the resulting surface is visible in Figure 7.10a. The structure of the amorphous diamond coating and the individual crystals can be seen in Figure 7.10b. The crystal structure can be seen to be fairly rounded which is indicative of diamond production using CVD.

Various amorphous diamond coating thicknesses were trialled. Similar to the alumina-zirconia coating described in Section 6.6, the coating thickness was limited by the sharp change in angle at the probe tip. It was preferable to increase the thickness of the coating as much as possible in order to provide the maximum abrasion resistance. However, it was observed that it was not possible to get the coating to adhere beyond a thickness of 5 μm. In order to ensure consistency, the coating thickness was specified at 2 μm.

As stated before, the thickness of the amorphous diamond coating is the key factor in determining the response time of the sensor. Work done by Penty Geraets et al. [102] shows that by modelling the system as a calorimeter, it is possible to define a rise time, τ_{rise} , which describes the time taken for the measured temperature to come within 1% of the mean temperature of the amorphous diamond coating. For a 2 μm

CVD diamond, the rise time is predicted to be approximately 0.1 μs

After application of the amorphous diamond coating, the probes were then calibrated using the method described in Section 6.4.

7.4.3 Calculation of heat fluxes from a shielded thin film gauge

The calculation of heat fluxes from a shielded thin film gauge is slightly more complex than the multi-layer system (described in Section 6.3.1) as it is the interface temperature being measured by the thin film gauge.

In order to calculate the heat flux from the thin film gauge under the amorphous diamond coating, the impulse response method (described in Section 6.3.2) was used. In order to provide the basis functions, $q_1(t)$ and $T_1(t)$, required to calculate the impulse response filter for the system, a 3-D FEA model was used. It should be noted that estimates for the thickness and thermal properties of the amorphous diamond coating from the design specification and data sheet were used in the 3-D FEA model.

7.4.4 Results of applying an amorphous diamond coating

The purpose of applying the amorphous diamond coating was to increase the robustness of the sensor during the engine cycle. A number of amorphous diamond coated samples were placed in the single cylinder test bench (described in Section 7.2).

It was observed that, on average, the uncoated ceramic samples failed midway during the second set of testing (with each set lasting 30 seconds), with the first set of testing including a cold start condition. This was cumulatively 45 seconds of testing during the firing condition.

With the amorphous diamond coating applied, the samples would, on average, fail towards the end of the third set of testing (with the first set of testing including a cold start condition). It is estimated that this was approximately 80 seconds of cumulative testing during the firing condition. Figure 7.11 shows the heat flux measured in the first 5 cycles of a cold start. The profiles observed can be considered very similar to the values of the pure ceramic substrate (seen in Figure 7.5b).

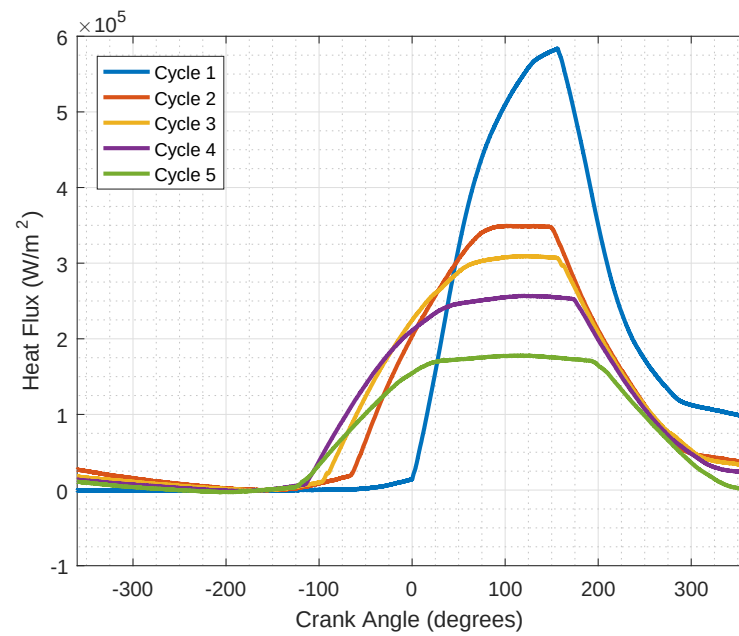


Figure 7.11: Measured heat flux during cold start condition using amorphous diamond coated thin film gauge

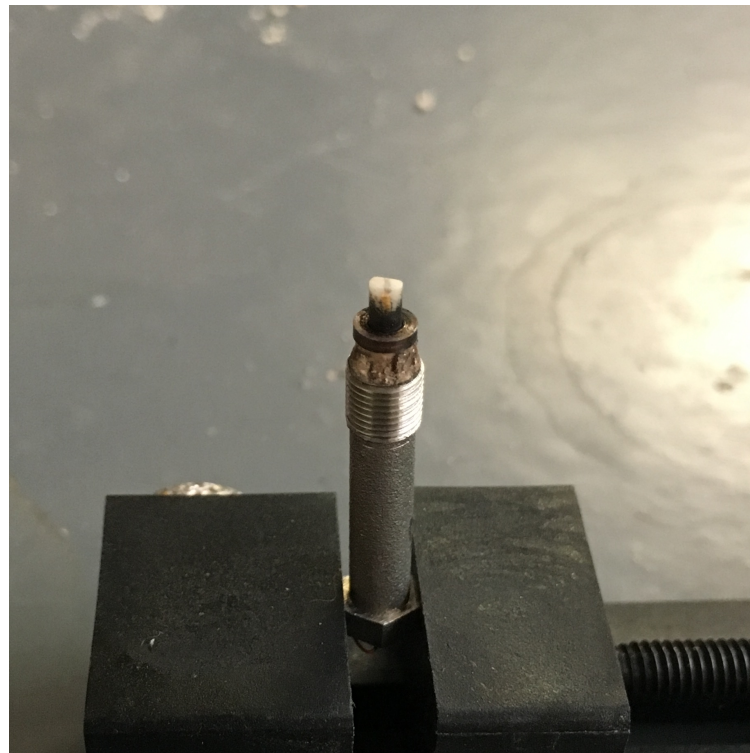


Figure 7.12: Amorphous diamond coated probe after testing

The values can be seen to be lower than those observed using the pure ceramic substrate. This was a common pattern across all conditions as the amorphous diamond coated gauges appeared to under-report the heat fluxes measured by the pure ceramic substrate. This indicates inaccuracies in the calculation of the thermal properties and thickness of the amorphous diamond coating. However, the similarities in the profile of the results measured using the pure ceramic gauges and amorphous diamond coated gauges respectively indicate that this is an issue that could be solved through accurate characterisation of the amorphous diamond coating.

After removing the thin film gauge from the single cylinder test bench, it was observed that the diamond coating had been removed from the the probe tip (but was still present further down the probe shaft). This is shown in Figure 7.12. This extension of the sensor life, coupled with the clear abrasion of the probe surface, indicates that the amorphous diamond coating does provide some abrasion resistance. However, the thickness and adherence of the amorphous diamond coating that is achievable on the current probe geometry limits the scope of any robustness improvements.

7.5 Summary of experimental findings

The work in Chapters 6 and 7 examined the use of thin film gauges for in-cylinder heat transfer experiments. Two methods of manufacturing thin film gauge sensors were investigated, which were the two layer and single layer systems. The use of a two layer system (with Inconel 625 and vitreous enamel) has been demonstrated [91] [94] but was hindered by the difficulty of manufacturing intricate geometries. The use of direct metal laser sintering (DMLS) to create complex geometries was trialled and the application of thin film gauge sensors on both vitreous enamel and alumina-zirconia coatings were shown to be successful.

It was shown that the thin film gauges manufactured using the ceramic substrate were able to meet the frequency response specification and mounting requirements outlined in Chapter 6. Noise artifacts were observed during the response time testing of the thin film gauges and were thought to be caused by interference during the switching of the power supply. The cause of these noise artifacts could be confirmed by an additional set of experiments, where it is verified that the noise artifacts are

still present when the incident radiance on the thin film gauge is blocked (but where the power supply is still switched on and off).

The sensors were then tested in a air-cooled single cylinder test facility in order to investigate the robustness of the solution in a representative environment.

The results from the single cylinder test facility were compared to empirical heat transfer models that are widely used in industry. It was observed that the agreement between the measured results and the empirical models mirrored those reported in the literature, such as good agreement with the Hohenberg model at peak values. The Woschni model was found to consistently over-estimate the heat transfer in the system. The profile of the heat flux values throughout the engine cycle were found to differ from both the empirical models. A possible explanation was that gradual removal of the platinum sensing circuit could lead to erroneously high temperature measurements.

Experiments conducted using the single cylinder test facility illustrated the difficulties in applying thin film gauge sensors to in-cylinder heat transfer experiments. The manufactured sensors were found to have robustness issues which was thought to be caused by abrasion of the sensing circuit. An amorphous diamond coating was trialled in order to increase the robustness of the sensor. The coating was successfully applied over the top of the sensing circuit with minimal changes to the electrical characteristics. The amorphous diamond coating was found to increase the robustness of the sensor, but the coating thickness (approximately 2 μm) and subsequent improvements to the gauge robustness was severely limited by the geometry restrictions imposed.

Further work should include the use of the amorphous diamond coating on a less restrictive geometry that allows for a thicker coating to be applied, which would allow a more rigorous evaluation of any improvements to sensor robustness. An alternative approach to protect the sensor in the engine cylinder would be to use a fast insertion technique in order to only subject the sensor to the in-cylinder environment for short durations.

Measurement of the heat flux values using the amorphous diamond coating in the single cylinder test facility was observed to be similar to the uncoated sensor for the cold

start condition. With the minimal impact on the sensing circuit performance, combined with the observed improvement in gauge robustness, the use of an amorphous diamond coating allows the possibility of manufacturing robust thin film sensors that are able to function in harsh operating conditions.

Chapter 8

Optimising Spark Advance using Wall Temperature

8.1 Introduction

A key factor in achieving maximum performance from an internal combustion engine is the initiation of the combustion process at the optimal point during the engine cycle. A key factor in achieving this is the spark advance timing used to initiate the combustion of the air-fuel mixture inside the cylinder.

For consumer vehicles, spark advance is generally controlled using an open loop control system implemented within the engine control unit (ECU) and is calculated using sensor outputs, located at various points in the power unit. The spark advance for a cycle is then calculated from a look up table by using these sensor values as input parameters [67].

8.2 Proposed System for Predicting Cylinder Wall Temperature

The inner cylinder wall temperature is dominated by the influence of two main parameters which are the heat flux into the inner cylinder wall (caused by the piston movement and combustion process) and the heat rejected to the coolant jacket on the outer face. Cylinder wall temperature is a useful parameter when calculating the optimum spark advance. However, it is difficult to measure during engine running due to the harsh conditions within the cylinder (see Section 6.2).

The system proposed in the present research aims to implement a method of calculating the cylinder inner wall temperature from the predicted heat flux into the cylinder wall and measurements of the coolant temperature. High cylinder wall temperatures are a key contributor to engine knock, which has the potential to cause damage to the cylinder [108]. Therefore, it is potentially advantageous to use wall temperature predictions in the calculation of spark advance in order to avoid frequent engine knock. This was particularly important for our industrial partner, whose engines run as close to the knock limit as possible to extract maximum performance.

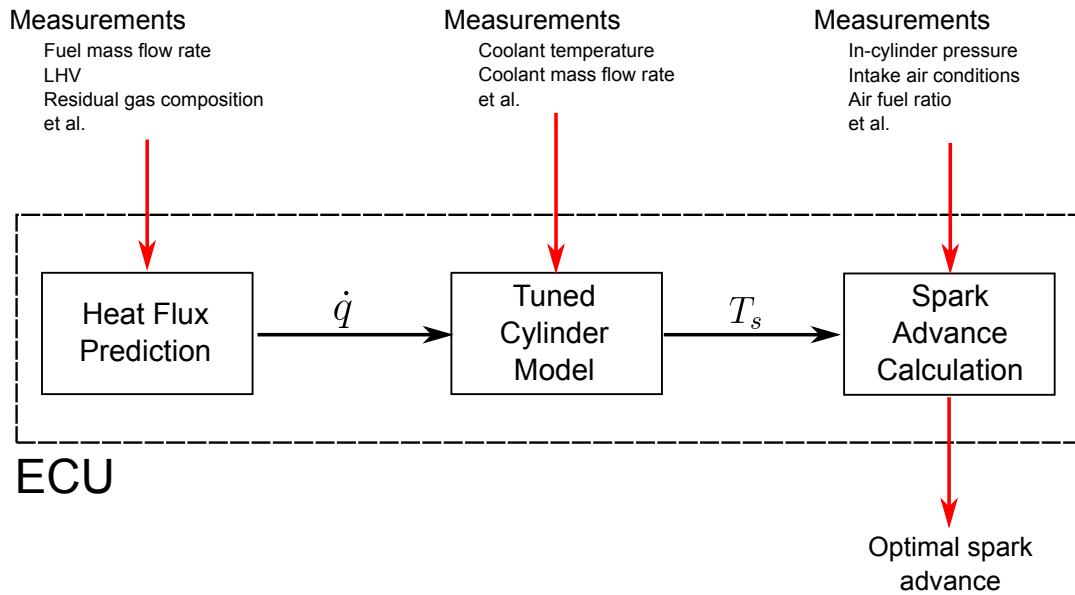


Figure 8.1: Schematic of Proposed System

Figure 8.1 shows a schematic of the proposed system. As the spark advance calculation for modern engines is controlled in real-time by the engine control unit, a key factor in implementing the proposed system is the time in which the calculation can be performed. Any prediction for the cylinder inner wall temperature (using information from the previous cycle) must be calculated during the intake and compression phases of the next cycle, such that the cylinder wall temperature can be used as an input to calculate the spark advance during the combustion phase.

There are two components to the proposed system:

1. Calculation of Heat Flux Profile

The input to the proposed system is the heat flux profile from the previous engine cycle. Calculation of the heat release from the engine cycle is hard to directly measure (see Chapter 6), but can often be re-constructed from pressure data [61]. For engines without feedback from a pressure transducer, other measurable parameters (such as engine speed and loading) can be used to predict the heat flux into the cylinder wall [60]. This model was implemented by our industrial partner.

2. Calculation of the Cylinder Wall Temperature

This component of the research is focused on the development of a numerical model that aims to calculate the cylinder wall temperature in real time. The model utilises two empirically tuned methods of solving the partial differential equations that govern the heat transfer within the cylinder wall. The system aims to take the predicted heat flux into the inner cylinder wall from the previous engine cycle and coolant temperature in order to predict the cylinder inner wall temperature. The effect of these two thermal inputs are separated and solved independently, before being superposed to predict the overall system response.

8.3 Superposition of Combustion and Coolant Cases

The proposed system is focused on the development of an empirically tuned cylinder model which uses the predicted heat flux into the cylinder wall and the measured coolant temperature in order to predict the cylinder wall temperature. As described in Section 8.2, the proposed tuned cylinder model is based on the superposition of two individual solutions to the 1-D Fourier equation to calculate the overall cylinder wall temperature.

As described in Section 8.2, a key factor in the implementation of the proposed system is the speed in which the cylinder wall temperature can be predicted. 3-D finite element analysis (FEA) simulations represent the highest fidelity method of predicting the cylinder wall temperature given the aforementioned boundary conditions. However, the computation time required to run a full 3-D FEA simulations makes this approach infeasible for a real-time application. Thus, alternative methods

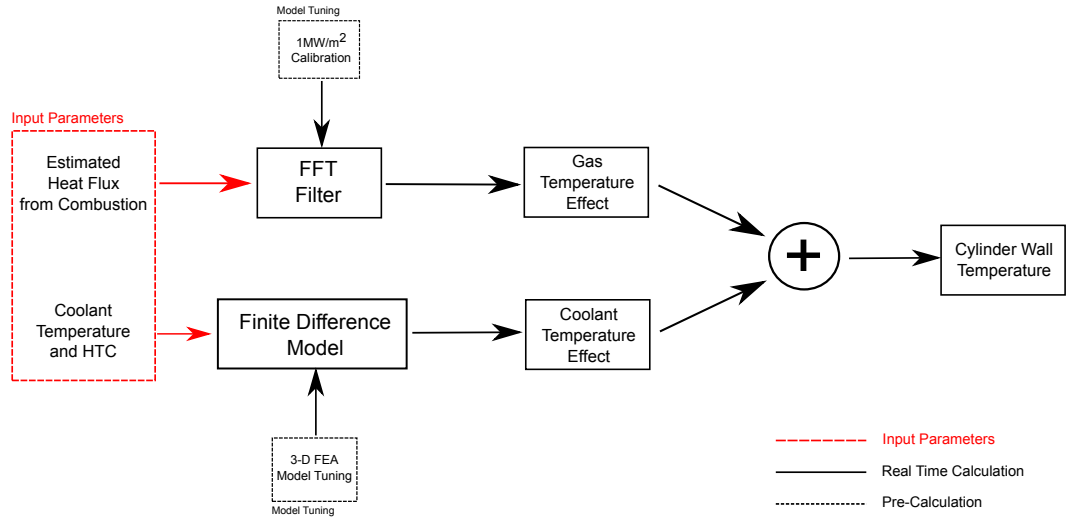


Figure 8.2: Superposition of combustion and coolant heat flux cases

were sought by the author to reproduce the accuracy that would be produced by a 3-D FEA simulation within a much shorter time span. The approaches to solve each system are outlined in Figure 8.2.

The system is split into the following two cases (defined as Case A and Case B) which are superposed to predict the resultant cylinder temperature (defined as Case C for the general case):

1. Heat Released from Combustion (Case A)

The effect of the heat flux into the cylinder wall (resulting from the piston motion and combustion process) was calculated using the fast Fourier transform method described by Oldfield [95].

2. Heat Rejection into Coolant (Case B)

The heat rejected into the coolant is an important factor in predicting the surface wall temperature. The effect of the coolant temperature on the inner cylinder surface can be calculated using a finite differing approach due to the small and slow changes in the coolant temperature.

8.3.1 Boundary conditions for superposition

In order to superpose the results from the two aforementioned cases, the boundary conditions for each case must be carefully chosen to provide bounded solutions to the governing equations while also giving a correct prediction to the actual conditions when superposed (shown in Figure 8.3).

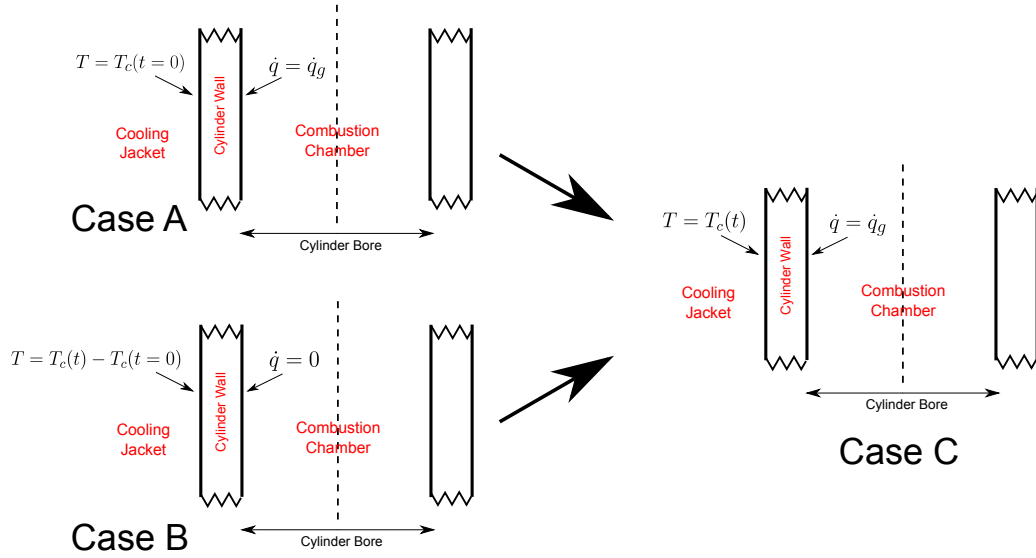


Figure 8.3: Boundary conditions for separate tuned cylinder model cases

If the system is reduced to a 1-D slice through the cylinder wall, the system is reduced to two boundary conditions on the outer and inner face. As a temperature boundary condition is required when solving each case to produce a bounded solution, an arbitrary temperature value must be defined on the other surface when solving for Case A as the only boundary condition available is gradient based (set by the heat flux release from the combustion process).

In order to form the correct boundary conditions in the superposed condition (Case C), the constant temperature boundary condition in Case A is subtracted from the relevant coolant temperature boundary condition in Case B. This allows the correct result when Case A is superposed on Case B to form the overall solution to the system (Case C). The outer wall of the cylinder is considered to be at the same temperature

as the coolant.

The required boundary conditions to give the correct predictions for the overall system response are the time varying heat flux from the engine cycle on the inner surface and the time varying coolant temperature on the outer surface at the cylinder. A summary of the boundary conditions for each case is described in Table 8.1.

Table 8.1: Summary of boundary conditions for superposition

Case	Inner Surface Boundary Condition (Temperature)	Outer Surface Boundary Condition (Heat Flux)
Case A	$T_c(t = 0)$	$q_g(t)$
Case B	$T_c(t) - T_c(t = 0)$	0
Case C	$T_c(t)$	$q_g(t)$

8.4 Use of the FFT Method to Predict Cylinder Wall Temperature

8.4.1 Introduction to the FFT method

The impulse response method, set out in Section 6.3.2, describes the steps necessary to predict the heat flux response from a surface temperature signal. In the present research, the proposed model seeks to derive a surface temperature response from a heat flux input. This is based on the impulse response method described by Oldfield [95] and is outlined in Section 6.3.2.

In Equation 8.1, the surface temperature can be calculated from the predicted heat flux by convolving the impulse response function of temperature to heat flux with the predicted heat flux.

$$T(t) = h_{q-T}(t) * q(t) = \int_{-\infty}^{\infty} h_{q-T}(\tau) q(t - \tau) d\tau \quad (8.1)$$

As described in Section 6.3.2, Equation 8.1 is simpler to solve in the discrete domain. Applying the z-transform to the discrete form of Equation 8.1 leads to the discrete convolution operation shown in Equation 8.2.

$$T_1(z) = H_{q-T}(z)Q_1(z) \quad \text{or} \quad H_{q-T}(z) = \frac{T_1(z)}{Q_1(z)} \quad (8.2)$$

After the z-transform is applied, the partial function, $\Delta(z)$, can be used (as in Section 6.3.2) to calculate the infinite impulse response function from a known pair of heat flux and surface temperature profiles (T_1 and Q_1) as in Equation 8.3.

$$H_{q-T}(z) = H_{q-T}(z)\Delta(z) = \frac{T_1(z)}{Q_1(z)}\Delta(z) \quad (8.3)$$

8.4.2 Calculation of the impulse response function using FEA simulations

In order to use the method set out in Section 6.3.2, a pair of known heat flux and surface temperature profiles is required to solve Equation 8.3 for the discrete impulse response function, $H_{q-T}(z)$. The method outlined by Oldfield [95] uses an analytical method, reported by Doorly and Oldfield [94], to solve the 1-D Fourier equation for a known step change in heat flux and a semi-infinite boundary condition for either a single layer or multi layer system.

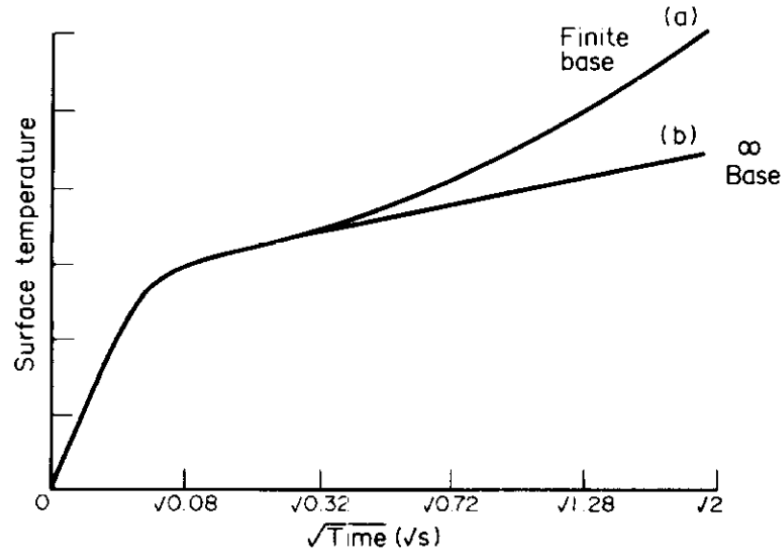


Figure 8.4: Example comparison between a) finite and b) semi-infinite boundary conditions for 1-D Fourier equation solutions [94]

The analytical method has been shown to give good results for traditional thin film gauge applications where the 1-D semi-infinite assumption is valid. However, the cylinder wall cannot be considered to be semi-infinite due to the thickness of the cylinder wall and the use of the temperature boundary condition on the outer wall (see Figure 8.4).

Although it is possible to use an analytical solution to calculate resultant surface temperature response for this set of thermal boundary conditions, the solution process becomes increasingly complex with both additional layers of material and finite temperature boundary conditions. Therefore, an alternative method of calculating the known heat flux and surface temperature pair using a full 3-D finite element analysis (FEA) simulation is investigated.

As previously mentioned, running a 3-D FEA simulation represents the highest fidelity method of predicting the cylinder inner wall temperature given the aforementioned boundary conditions. However, the computational expense involved ruled this method infeasible for real-time applications. As the impulse response function only needs to be calculated once for a given system (assuming no changes in geometry or sampling frequency), a full 3-D FEA simulation can be used to calculate the required surface temperature response from the given set of thermal boundary conditions. This is possible as the impulse response function is pre-calculated and so any computational expense and time taken does not need to be taken into account when considering the real-time operation speed of the method.

An advantage of using a 3-D FEA methods is the ability to introduce and account for geometrical complexity (such as curvature) and multi-layer effects with relative ease. As these features can be easily modelled using modern MCAD packages, it is then fairly straightforward to account for these features in the 3-D FEA simulation.

8.5 Prediction of Coolant Effects using Finite Differencing

8.5.1 Effect of the coolant on cylinder wall temperature

The outer surface of the cylinder in an internal combustion engine is often cooled by a coolant jacket in order to dissipate in the cylinder wall. Due to the very large heat transfer coefficient at the interface between the outer surface of the cylinder and the coolant jacket, this effect can be represented by a temperature boundary condition on the outer surface of the cylinder.

As outlined above, a constant temperature boundary condition is required to find the solution of the heat flux boundary condition, calculated using the fast Fourier transfer method (see Table 8.1). In order to achieve the correct boundary conditions when the effects of the coolant temperature and heat flux into the inner surface of the cylinder are superposed, the boundary conditions used to calculate the effect of the coolant temperature is offset.

Using this offset temperature boundary condition, the effect of the time-varying coolant temperature is calculated using a finite difference method.

8.5.2 Calculation of the temperature profile through the cylinder wall

The effect of changes in the coolant temperature, which is the same as the outer cylinder wall temperature, is captured by calculating the temperature profile through the cylinder wall. As before, if curvature and lateral conduction effects can be neglected, the system can be reduced to an application of the 1-D Fourier equation as shown in Equation 8.4.

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \quad (8.4)$$

Explicit finite differencing can be used to solved partial differential equations by approximating the derivative terms using finite differences in both time and space. As applied to Equation 8.4, the first order and second order derivatives in time and space

respectively can be approximated using Equations 8.5 and 8.6 where T_j^n represents the temperature at time point n and at spatial coordinate j .

$$\frac{\partial T}{\partial t} = \frac{T_j^{n+1} - T_j^n}{\Delta t} \quad (8.5)$$

$$\frac{\partial^2 T}{\partial x^2} = \frac{T_{j-1}^n - 2T_j^n + T_{j+1}^n}{(\Delta x)^2} \quad (8.6)$$

Substituting Equations 8.5 and 8.6 into Equation 8.4 leads to Equation 8.7. This can be simplified as the material thermal diffusivity (α), and step sizes (Δt , Δx) can be combined into the finite-difference form of the Fourier number, λ . Rearranging to solve for the temperature at the next time step T_j^{n+1} results in Equation 8.8.

$$\frac{T_j^{n+1} - T_j^n}{\Delta t} = \alpha \frac{T_{j-1}^n - 2T_j^n + T_{j+1}^n}{(\Delta x)^2} \quad (8.7)$$

$$T_j^{n+1} = T_j^n + \lambda (T_{j-1}^n - 2T_j^n + T_{j+1}^n) \quad (8.8)$$

$$\text{where } \lambda = \frac{\alpha \Delta t}{(\Delta x)^2} \quad (8.9)$$

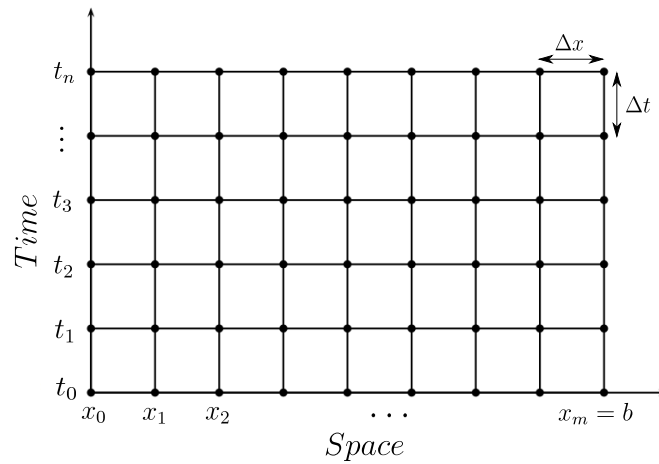


Figure 8.5: Finite difference grid

The cylinder wall can be represented as the grid shown in Figure 8.5 where the cylinder wall is considered to have a thickness b . The horizontal axis represents the cylinder wall separated into m discrete spatial points and the vertical axis represents the solution of each of these spatial points for n discrete time points. The spacing of these points in time and space are represented by the Δt and Δx points respectively. The temperatures throughout the wall are taken to be cell-centred values.

A disadvantage of using the explicit finite difference method is that it is not unconditionally stable, with the stability dependent on the λ value chosen. This places constraints on the grid spacing, Δt , and time stepping, Δx , used in the model. Incropera et al. indicates that the stability criterion for the explicit finite difference method (as applied to interior nodes) is $\lambda \leq \frac{1}{2}$ [73].

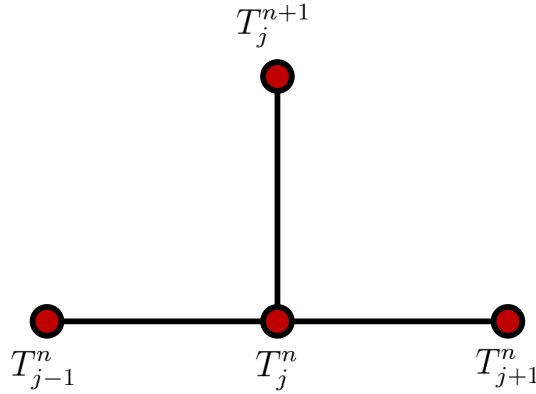


Figure 8.6: Explicit finite difference scheme stencil

The application of Equation 8.8 can effectively be represented by the explicit scheme stencil shown in Figure 8.6. This can be overlaid on any point in the grid (shown in Figure 8.5) to show the points in space and time that are relevant to the calculation of the temperature at the next time step T_j^{n+1} .

8.5.3 Boundary conditions for finite difference model

Applying the explicit difference stencil (shown in Figure 8.6) to the grid (shown in Figure 8.5) requires specific handling at the cells adjacent to each of the cylinder walls

between the end of the cycle and the next combustion phase. This is illustrated in Figure 8.8, which shows a heat flux trace for six cycles.

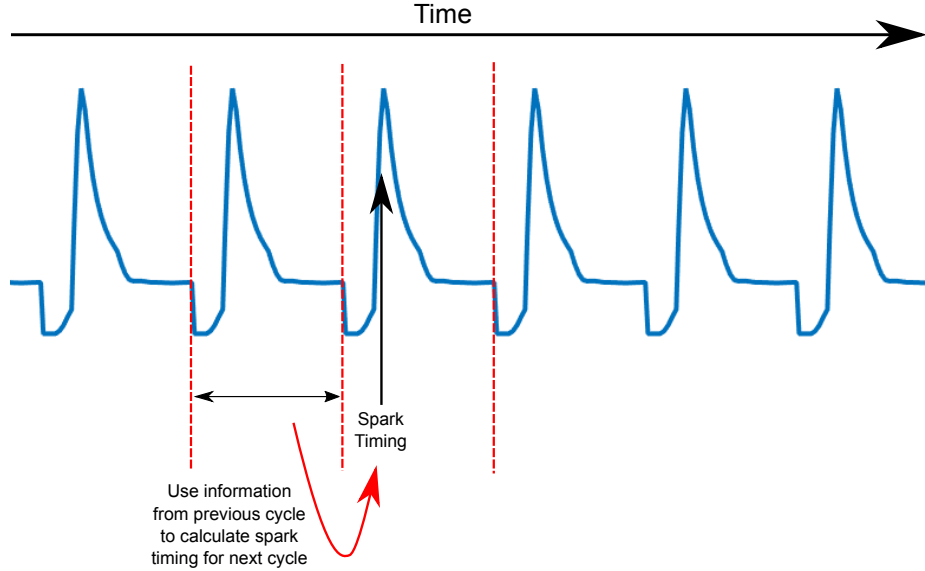


Figure 8.8: Overview of real-time implementation

In order to implement the model in real-time it is necessary to differentiate the requirements from the implementation described in Sections 8.4 and 8.5.

The effect of the coolant temperature can readily be predicted in real time using the finite difference method. As the effect of the coolant temperature is calculated from the data at the previous time step and the coolant temperature at the current time step, the sequential nature of this calculation can be used in real-time with minimal adaptations.

8.6.2 Implementation of the FFT method in real-time

The original FFT method, as proposed by Oldfield [95], was designed for use at the post-processing stage with the full surface temperature profile available in order to calculate the heat flux applied.

As described in Section 6.3.2, the FFT methods is used to calculate the output of a LTI system by filtering the input signal using the impulse response function. The impulse response function uses the historical values of the input signal to calculate the output value at the current time step using a convolution operation. This impulse

response function is initially calculated using a known pair of input and output signals for the system under consideration.

For the given system of the engine cylinder, this known pair of surface temperature and heat flux profiles is generated using a 3-D FEA model. A step change in heat flux applied at $t = 0$ is used as the calibration signal which generates a continuous surface temperature response (calculated using a 3-D FEA simulation). An example of a surface temperature response to a constant heat flux input can be seen in Figure 8.9.

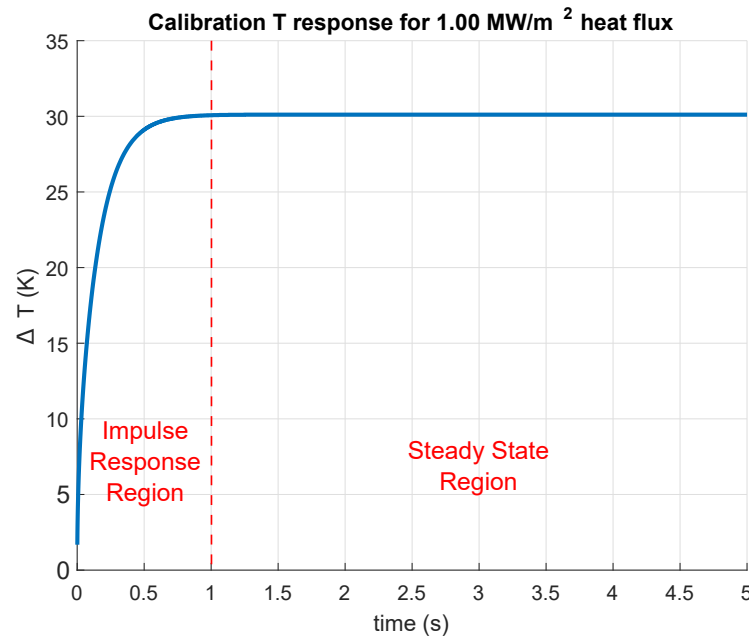


Figure 8.9: Example calibration of cylinder wall system

The surface temperature response shown in Figure 8.9 can broadly be separated into two phases. As described in Section 8.4, the solution shown in Figure 8.9 is sampled at a set frequency to calculate the impulse response function for the geometry and sampling parameters. The method outlined by Oldfield [95] calculates the infinite impulse response function (IIR) of the signal.

From Figure 8.9, it can be seen that the surface temperature response to the input heat flux signal doesn't change after a certain time interval. This indicates that the impulse response function, calculated using the method described in Section 8.4, tends

to zero which can be used to define the number of historical input values required to calculate the output at a given time step.

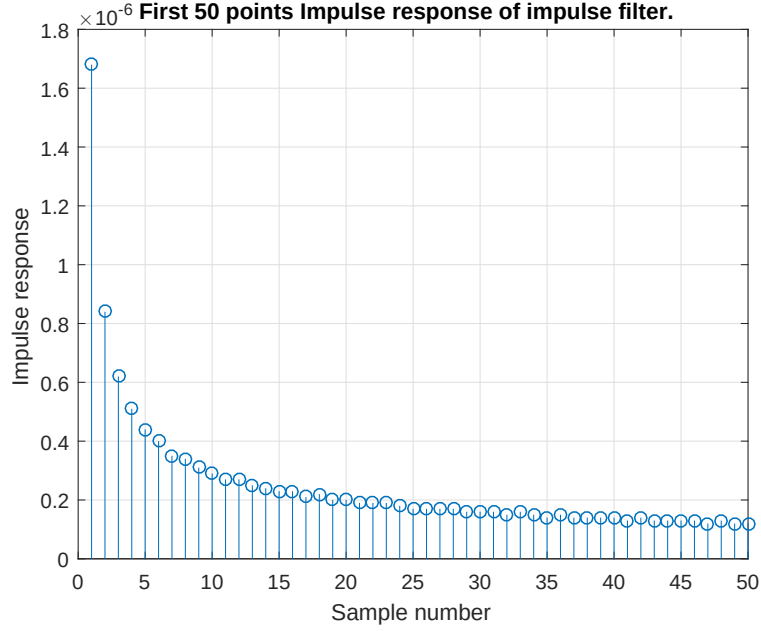


Figure 8.10: Example impulse response function for cylinder wall system

Figure 8.10 shows the first 50 points of the impulse response function generated using the surface temperature and heat flux profile pair shown in Figure 8.9 and represents the conversion of heat flux to surface temperature for the cylinder wall model through a discrete convolution operation with a heat flux input signal.

It can be seen in Figure 8.10 that the values of the impulse response function is asymptotic and reduces as the sample number increases. Beyond a certain point, the historical input value from the input signal will have negligible effect on the output value at the current time point. This can be defined in terms of the non-dimensional Fourier number (shown in Equation 8.10). A value of $Fo > 4$ was considered to be appropriate to ensure that the impulse response function was a sufficient length [109].

$$Fo = \frac{\alpha t}{L^2} \quad (8.10)$$

Using this information, it is possible to identify the number of discrete historical heat flux values that need to be stored in order to generate a surface temperature

prediction at the current time point. This defines the minimum required length of the impulse response function that is used to filter the input heat flux signal to calculate the surface temperature at the current time step.

8.6.3 Structure of FFT real-time implementation

When the zero-point location of the impulse response function has been identified, this indicates the number of historical heat flux values that must be stored in order to calculate the cylinder inner surface temperature at the current time point.

Figure 8.11 shows the method used to implement the FFT method in real-time. A first-in, first-out stack (FIFO) data structure is used to store the historical heat flux data that is filtered by the impulse response function to calculate the surface temperature at the given time step. The length of the FIFO stack is the same length as the impulse response function as the latter defines the number of historical values required to calculate the output at the current time step.

As described in Section 8.4, a known surface temperature and heat flux signal pair is generated using a 3-D FEA simulation. This result is then sampled at the required frequency to generate a discrete series that is used to calculate the impulse response function for the system at the given sample frequency.

It should be noted that the duration of the 3-D FEA simulation should be sufficient such that the impulse response function will reduce to a point where the effect of historical values will have negligible effect on the calculation of the output at the current point. This zero-point of the impulse response function is then used to define the minimum number of samples required to calculate the output at the given time point. This defines both the required length of the FIFO stack and the length of the impulse response function that needs to be stored from the original calibration simulation.

During engine running, it is assumed that a packet of data will contain N discrete samples containing the heat flux release and coolant temperature. This packet of N samples is added to the FIFO stack. The length of the stack is then compared to the length of the impulse response function to assess if there is any redundant data.

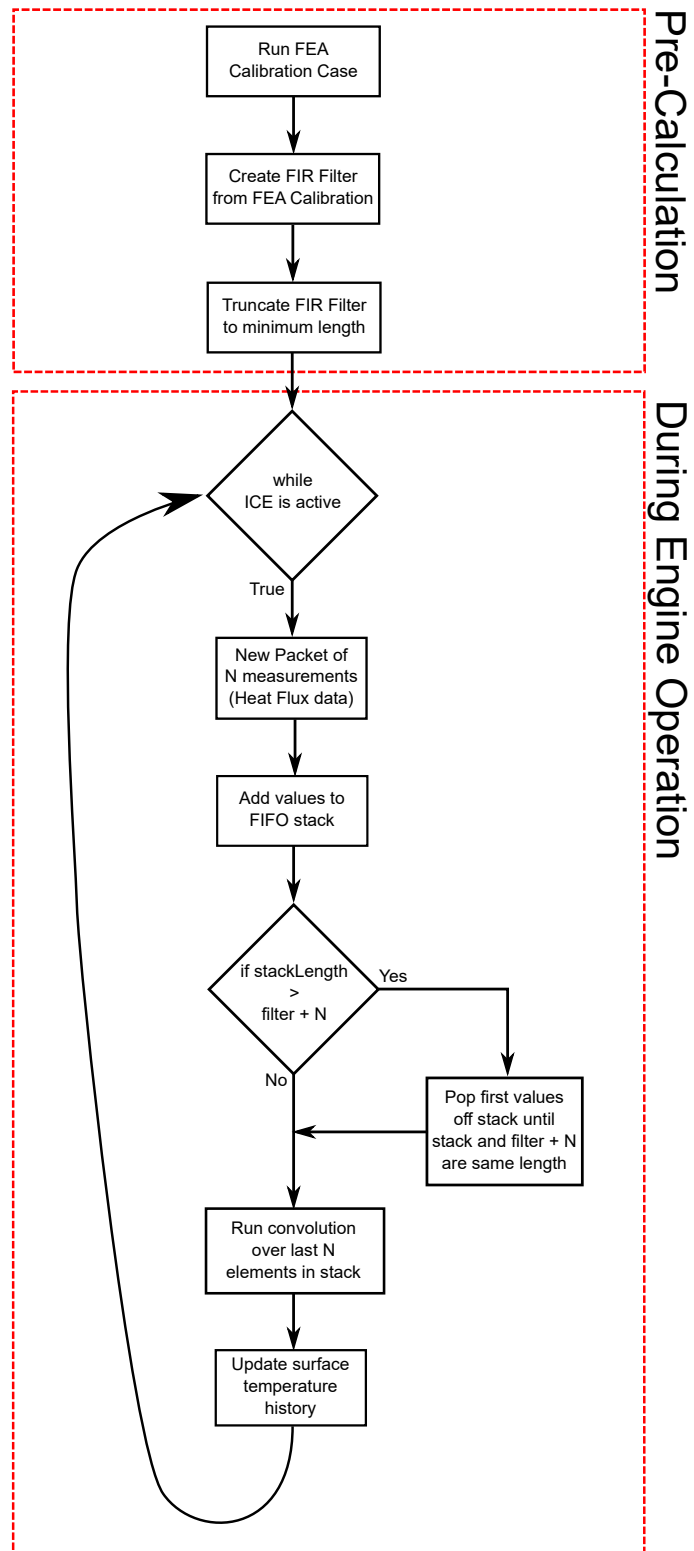


Figure 8.11: State diagram for real-time application of FFT Method

The stack needs to store samples equivalent to the length of the impulse response function in addition to the N samples from the new data packet. This is required as the convolution operation to calculate the surface temperature at the first time point in the new packet of data requires the historical data stored in the stack. Any historical data past this point is redundant and can be discarded.

The discrete convolution sum can then be calculated for the new elements in the stack to predict the surface temperature signal.

Chapter 9

In-Cylinder Surface Temperature Computational Results

9.1 1-D Conduction Model

9.1.1 Computational model description

In order to assess the accuracy of the proposed system, computational models were created and simulated using a 3-D FEA package to provide a benchmark against which to compare the performance of the proposed model. These computational models were also used to tune the proposed cylinder wall model.

The first model aimed to simplify the system to a straightforward 1-D conduction problem. A 100 mm^2 area square was extruded by 4.5 mm (shown in Figure 9.1) to represent a slice through the cylinder wall (assumed to be 4.5 mm thick). The base material was assumed to be aluminium with a $100\text{ }\mu\text{m}$ Nikasil coating on the combustion side face. Nikasil is a nickel silicon-carbide coating that is often applied to the internal combustion surface in order to reduce wear and friction on aluminium cylinders.

A transient simulation was run using the ANSYS Mechanical software package to investigate the system behaviour for 1 second with static timestepping set to 1 ms. This was adequate for the system to reach a steady state, required by the time marching algorithm for real time implementation (as described in Section 8.6).

It should be noted that the impulse response function must be calculated using the

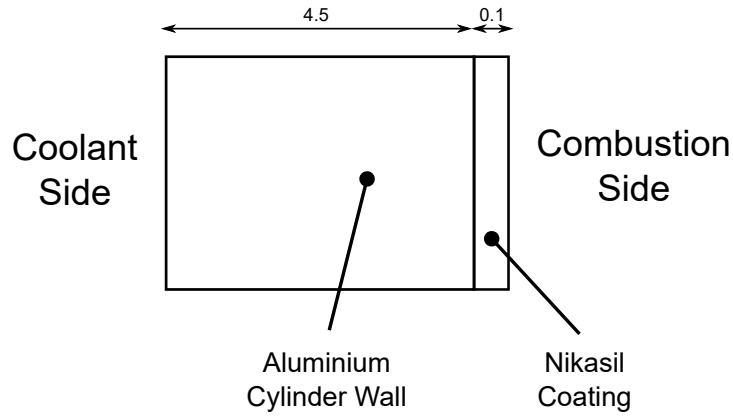


Figure 9.1: Outline of cylinder wall computational model

same sampling frequency as in the actual implementation. However, the sampling period is unrelated to the time step used during the 3-D FEA simulation as the results are re-sampled when designing the impulse response function. As described in Chapter 8, the time step in the 3-D FEA simulation must be sufficiently short to accurately model the system response to the input signal.

Table 9.1: Summary of boundary conditions for superposition

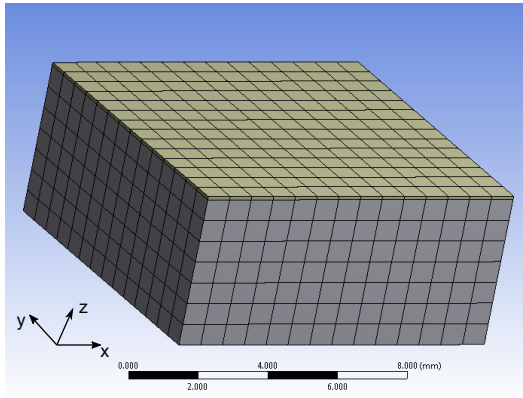
Case	Inner Surface Boundary Condition (Temperature)	Outer Surface Boundary Condition (Heat Flux)
Case A	403.15 K	$q_g(t)$
Case B	$T_c(t) - 403.15$ K	0
Case C	$T_c(t)$	$q_g(t)$

Table 9.1 describes the boundary conditions applied to each of the cases. For the constant coolant condition (Case A), a constant temperature condition of 403.15 K was imposed on the coolant side. The constant temperature boundary condition is then used as an offset when calculating the effect of the time-varying coolant temperature on the combustion side with an adiabatic heat flux condition imposed on the combustion face (Case B). The two cases can then be superposed to predict the surface temperature response subject to the overall system's thermal boundary conditions.

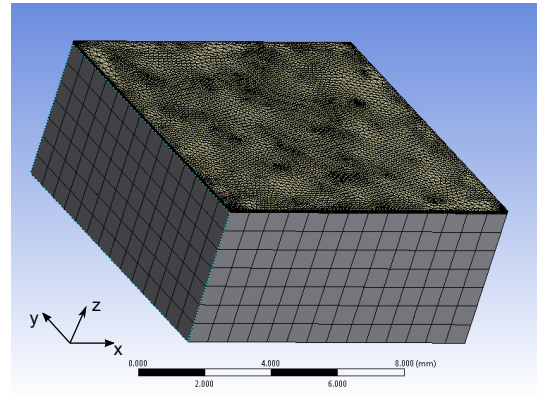
9.1.2 Mesh independence study

The effect of the mesh density on the results of the 3-D FEA simulation was examined at the beginning of this investigation.

The coarse mesh comprised 1768 elements (Figure 9.2a) whilst the fine mesh consisted of 36435 elements (Figure 9.2b).



(a) Coarse mesh for cube geometry



(b) Fine mesh for cube geometry

A refinement parameter was set on the Nikasil coating part for the fine mesh resulting in the coarse mesh comprising 1 layer of cells in the z-direction for the Nikasil coating, with the fine mesh comprising 3 layers of cells. The mesh density in the x-direction and y-direction is not important as only 1-D conduction through the domain is assumed (as the boundary conditions are symmetrical in both the x-direction and y-direction). Corner effects are neglected due to the use of adiabatic conditions on all faces apart from the top and bottom surfaces.

A 1MW/m^2 heat flux step was applied at $t = 0$ on the combustion side surface and the results compared for both meshes. A comparison between the surface temperature responses is shown in Figure 9.3. It can be seen that there are minimal difference between the two results, with an average error of 0.033 K across the entire simulation (1000 time steps at 1 ms intervals). From this result, the coarse mesh was used for the ensuing calculations in the interests of computational speed.

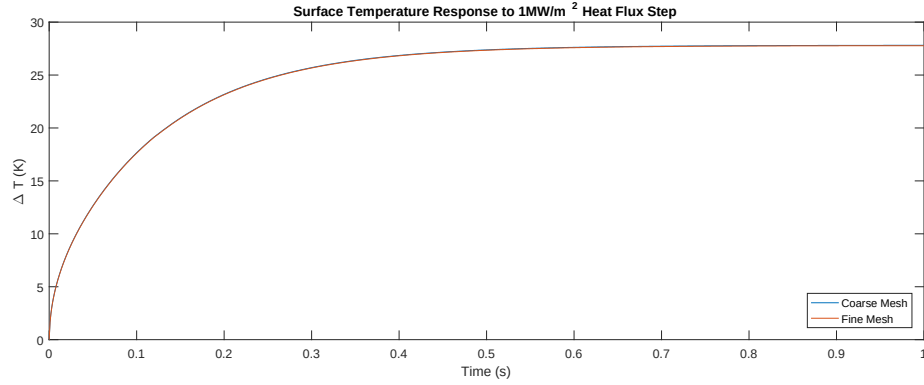


Figure 9.3: Comparison of results between coarse and fine computational meshes

9.1.3 FFT calibration error sensitivity

A key component of this system is the use of 3-D FEA simulation results to build the impulse response function (as outlined in the method contained in Section 8.4.2). As the result from the 3-D FEA simulation is re-sampled to match the system sampling frequency when designing the impulse response function, the time stepping used in the 3-D FEA simulation must be sufficient to accurately represent the system response.

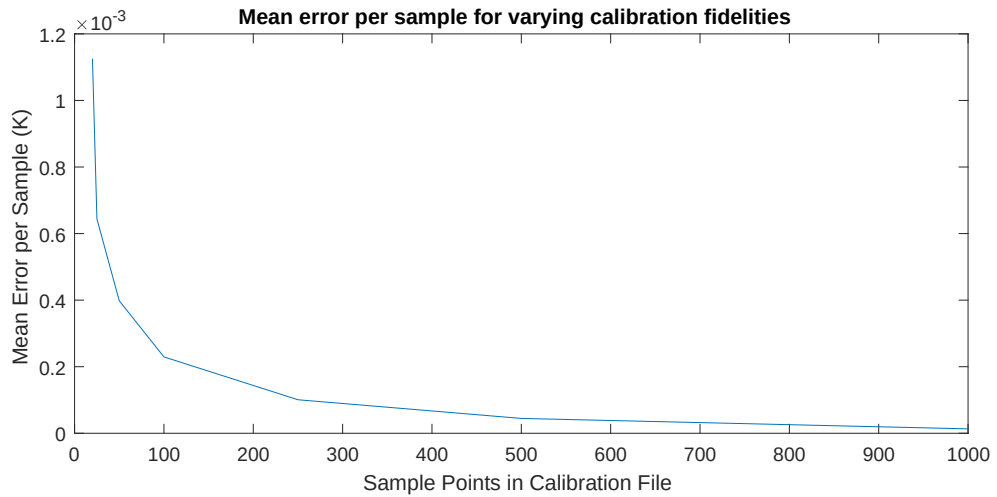


Figure 9.4: Resultant error from varying calibration fidelities

The resulting impulse response function is then used with the input heat flux to predict the surface temperature response. This is directly compared with the results from a datum 3-D FEA simulation with a time step of 1 ms (corresponding to a 1 kHz sampling frequency).

Figure 9.4 shows the mean error per sample of the resulting system as the time stepping of the 3-D FEA simulation is varied. The simulation consisted of a 1 second duration analysis, with re-sampling for the impulse response function at 1 kHz. The error cannot reduce when the re-sampling rate becomes lower than the time stepping in the 3-D FEA simulation as no interpolation occurs.

As expected, for a lower number of samples in the 3-D FEA simulation (corresponding to a large time step), the system response is not accurately captured by the 3-D FEA simulation, resulting in a large error when compared against the datum result. However, as the time stepping is reduced, the error observed can be seen to reduce as the 3-D FEA simulation result becomes closer to the datum response and the amount of required interpolation at the re-sampling stage is reduced.

9.1.4 Calculation of heat flux results

In order to calculate the impulse response function required to calculate the surface temperature response from the heat flux profile, a calibration case was required with a known surface temperature and heat flux pair applied to the system.

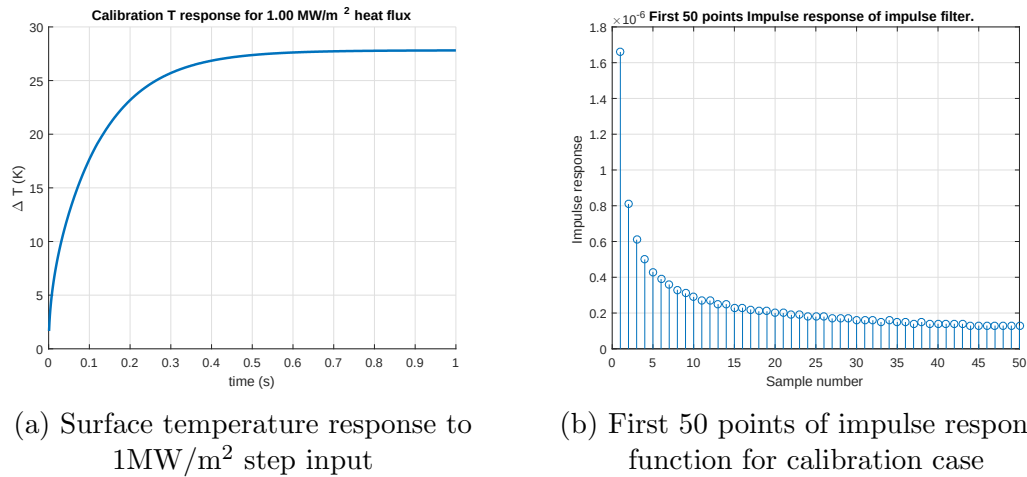


Figure 9.5: Surface temperature response to 1 MW/m² heat flux step and designed impulse response function

The original approach, as described by Oldfield [95], utilises the analytical solution to the Fourier equation with a semi-infinite boundary condition on the rear face. As

the boundary condition on the rear face for this system cannot be considered semi-infinite, the 3-D FEA model described in Section 9.1.1 was used to solve the system for the surface temperature response to a step change in surface heat flux.

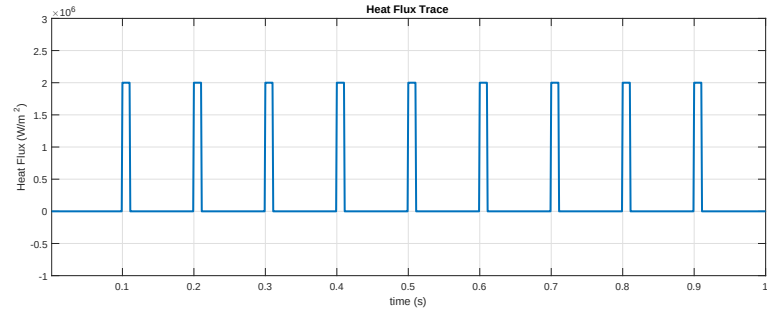
Any known heat flux input can be used to generate the surface temperature response using the 3-D FEA model. In this case, a 1 MW/m^2 step input at $t = 0$ was used (shown in Figure 9.5). As described in Section 8.4, a constant temperature boundary (403.25 K) was used on the coolant side.

Two heat flux profiles were applied to the geometry described in Section 9.1.1.

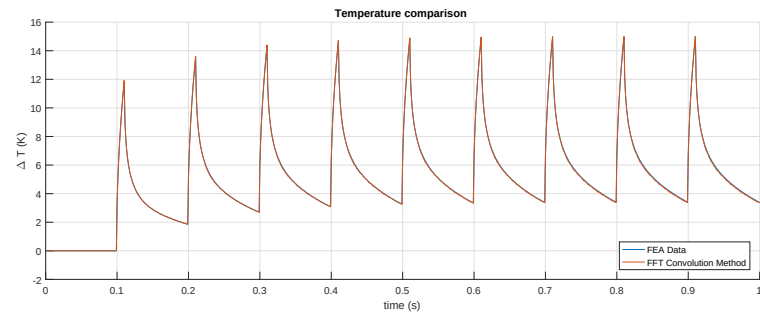
The first heat flux profile is shown in Figure 9.6a and shows a 2 MW/m^2 pulse train heat flux input signal imposed as a boundary condition on the combustion surface of the cube geometry. The resultant temperature profile predicted using the FFT convolution method is shown in Figure 9.6b. This predicted surface temperature response is then directly compared to the surface temperature response calculated using the 3-D FEA simulation. It can be seen that the temperature predictions for the combustion face made using the FFT method mirrors the results predicted the FEA method very closely, with agreement within 0.1 K across the duration of the simulation.

The second heat flux profile can be seen in Figure 9.7a and shows a varying sine wave input with amplitude 1 MW/m^2 which was imposed as a boundary condition on the combustion surface of the cube geometry. As before, the surface temperature response was predicted using the FFT convolution method and directly compared to the surface temperature response calculated using the 3-D FEA model (see Figure 9.7b). As in Figure 9.6b, there is very good agreement between the two methods, with the agreement within 1% at all time points.

It should be noted that the time required to run the 3-D FEA simulation for 1000 time points in order get the results shown in Figures 9.6b and 9.7b took around 4 hours to complete using 8 computation threads in ANSYS Mechanical. The calculation of the temperature predictions using the FFT method (post calculation of the filter, thus not including the initial calibration case) was run using a single computation thread and was completed in 0.000184 seconds.

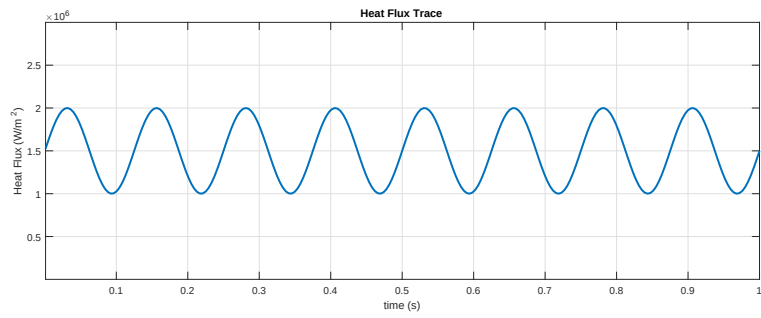


(a)

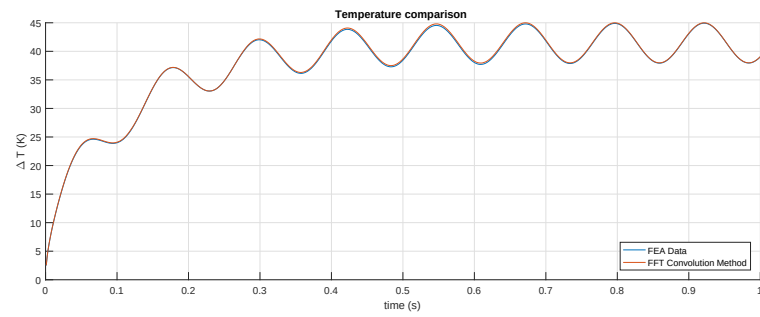


(b)

Figure 9.6: 2MW/m^2 pulse train heat flux input on combustion face



(a)



(b)

Figure 9.7: Sine wave heat flux input on combustion face

9.1.5 Calculation of coolant effects

In an operational engine, the coolant temperature is expected to vary with time (as represented in the boundary conditions described in Section 9.1.4). Therefore, it was important to be able to predict the effects of a time-varying coolant temperature on the combustion face of the engine cylinder. Using the technique described in Section 8.5, a finite difference scheme was used to calculate the coolant effect on the combustion face by calculating the temperature profile through the cylinder wall.

The coolant temperature profile used is shown in Figure 9.8. This was applied to the coolant face and the finite difference method was applied to calculate the resultant temperature on the combustion face. The temperature profile shown in Figure 9.8 is offset using the starting actual coolant temperature in order to aid the later superposition of the two solutions.

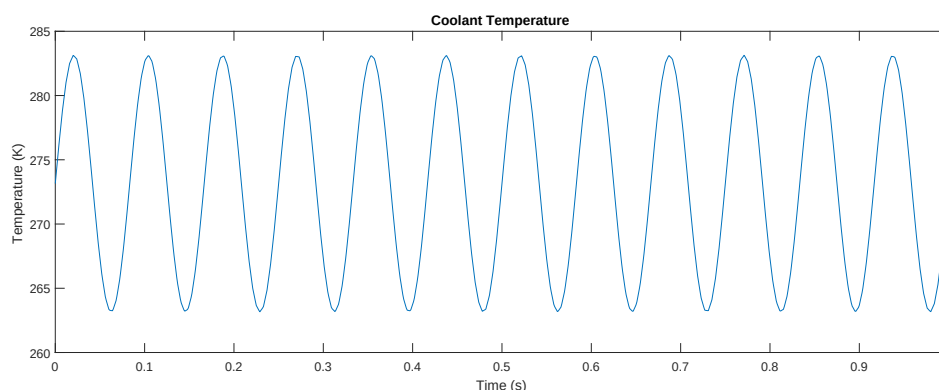


Figure 9.8: Time-varying temperature boundary condition applied to coolant face

As described in Section 8.5, linear interpolation and ghost cells are used to handle the boundary conditions of the finite difference model. As the system consists of two layers, the maximum cell spacing parameter was set by the thickness of the Nikasil coating. The different materials are accounted for by using Nikasil material properties for the single cell adjacent to the combustion surface and aluminium material properties for all others.

9.1.5.1 Model tuning using gradient descent

The stability and accuracy of the finite difference method is dominated by the choice of the finite difference Fourier parameter, λ (shown in Equation 9.1).

$$\lambda = \frac{\alpha \Delta t}{\Delta x^2} \quad (9.1)$$

As the choice of material thermal diffusivity, α , is set by the parameters in the 3-D FEA simulation, only the grid spacing, Δx , and time stepping, Δt , can be varied in order to tune the simulation. As varying the grid spacing effectively changes the geometry of the problem, the time step of the finite-difference of the model was used in order to tune the model.

A calibration case was generated by applying the coolant temperature profile shown in Figure 9.8 to the geometry described in Section 9.1. The combustion surface temperature was then predicted using the 3-D FEA model and compared to the combustion surface temperature predicted using the finite difference model using a default time stepping.

Table 9.2: Default material parameters for finite difference model

Property	Aluminium	Nikasil
Thermal diffusivity, α ($\frac{mm^2}{s}$)	93.83	2.99
Grid spacing, Δx (m)	0.0005	0.0001
Time stepping, Δt (s)	0.0005	0.0005
Fourier number, λ	0.1877	0.1495

Large discrepancies in both phase and magnitude were observed between the combustion surface temperature predicted using the finite difference model and the the 3-D FEA model. This made it clear that the default parameters for the model would not be acceptable if the 3-D FEA model was considered to be the accuracy benchmark for the proposed model.

A cost function, $J(\Delta t)$, was defined by calculating the mean squared error between the 3-D FEA model, $T_{s,FEA}$, and the finite difference model, $T_{s,FD}(\Delta t)$, for the N

timesteps in the re-sampled calibration case (see Equation 9.2).

$$J(\Delta t) = \frac{1}{N} \sum_{n=1}^N \left[T_{s,FEA} - T_{s,FD}(\Delta t) \right]^2 \quad (9.2)$$

A gradient descent method was then used in order to locate the minimum of the cost function (shown in Equation 9.2) by varying the time stepping parameter. An initial value of 0.0005 seconds was used as the starting value for the optimisation.

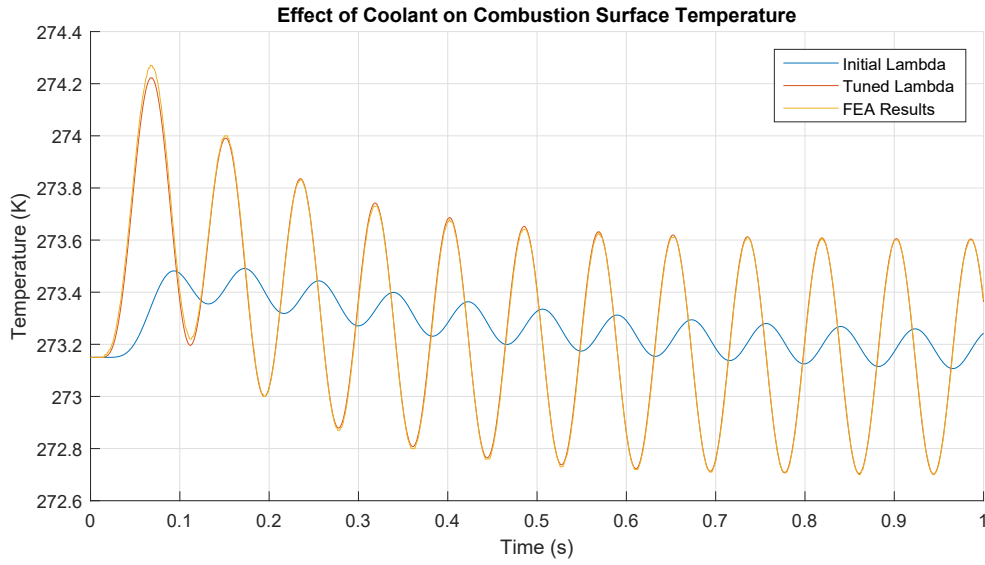


Figure 9.9: Effect of coolant temperature on combustion face for initial and tuned λ values compared with FEA prediction

The results of the optimisation stage can be seen in Figure 9.9 where the 3-D FEA results are shown compared to the finite difference model using both the initial default and optimised material property parameters. A large improvement in error reduction can be seen between the initial model and the model using the optimal time stepping, with agreement within 0.1 K for all time steps. The resulting parameters from the optimisation function can be found in Table 9.3.

9.1.6 Superposition of Results

The results in Sections 9.1.4 and 9.1.5 are then superposed to find a solution for the boundary conditions of the overall system.

Table 9.3: Optimum parameters for finite difference model

Property	Aluminium	Nikasil
Thermal diffusivity, α ($\frac{mm^2}{s}$)	93.83	2.99
Grid spacing, Δx (m)	0.0005	0.0001
Time stepping, Δt (s)	0.0011	0.0011
Fourier number, λ	0.3995	0.3289

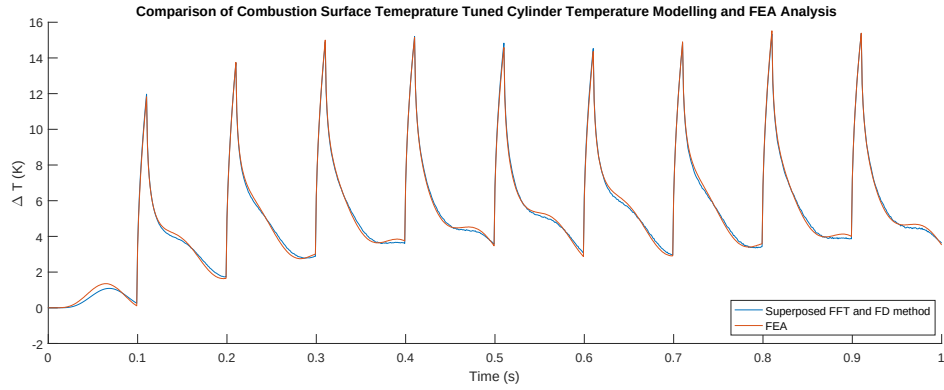


Figure 9.10: Comparison of tuned cylinder modelling technique with 3-D FEA simulation for square wave heat flux boundary condition

Figure 9.10 shows the superposed result for the heat flux boundary condition shown in Figure 9.6a imposed on the combustion surface. When compared to Figure 9.6b, the effect of the varying coolant can clearly be seen on the combustion surface temperature. Discrepancies can be observed in the regions of no heat flux addition, indicating that the finite difference method is the cause of the observed errors. However, it should be noted that both the phase and magnitudes of the temperature changes are well captured by the superposed models.

Figure 9.11 shows the superposed result for the heat flux boundary condition shown in Figure 9.7a as applied to the combustion surface. When compared to Figure 9.7b, the effect of the varying coolant can again be clearly identified. As in the previous example, the magnitude of the temperature fluctuations are well captured by the superposition of the FFT and finite difference methods.

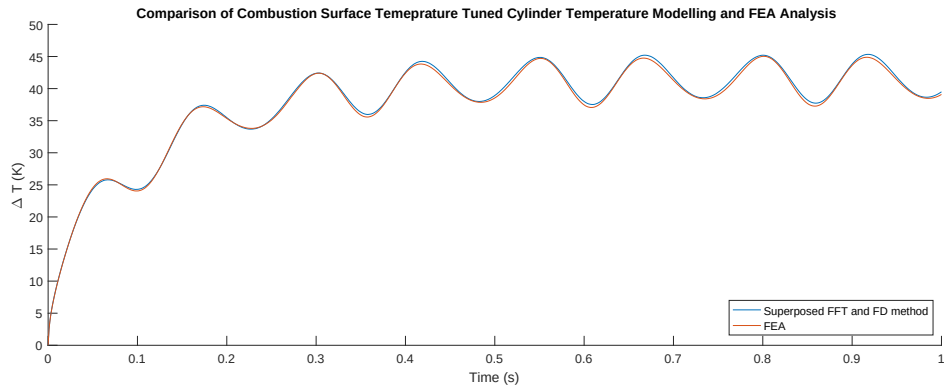


Figure 9.11: Comparison of tuned cylinder modelling technique with 3-D FEA simulation for sine wave heat flux boundary condition

9.2 Cylinder Cross Section Results

9.2.1 Computational model description

The approach outlined in Section 9.1 was extended to a 10 mm slice through the cylinder wall. The cylinder wall geometry was assumed to have an 80 mm bore with a 4.5 mm cylinder wall thickness. The base metal was assumed to be aluminium with a 100 μm Nikasil coating on the combustion side. The cylinder slice geometry was used in order to include any radial conduction effects. This builds on the work done in Section 9.1 which covered a multi-layer geometry.

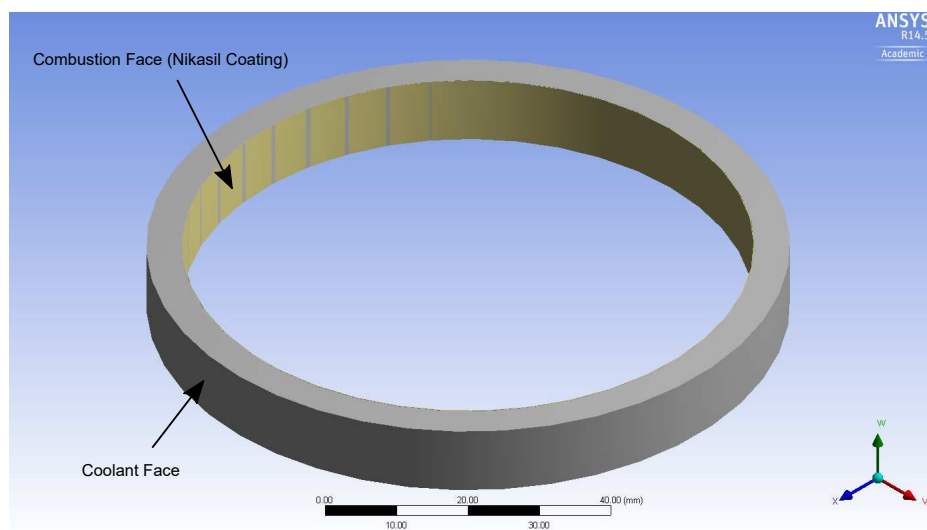


Figure 9.12: Overview of Cylinder Slice Geometry

As before, the surface temperature on the coolant side was assumed to be equal to the temperature of the coolant as a result of the heat transfer coefficient being sufficiently large. A transient solution of the finite element model was calculated using the ANSYS Mechanical package with 1ms time stepping. The mesh consists of approximately 80,000 elements.

9.2.2 Calculation of heat flux results

As in Section 9.1.4, a calibration case was run using the finite element model described in Section 9.2.1. A 1 MW/m^2 heat flux was applied to the combustion face with a constant temperature boundary condition (403.15 K) on the coolant face.

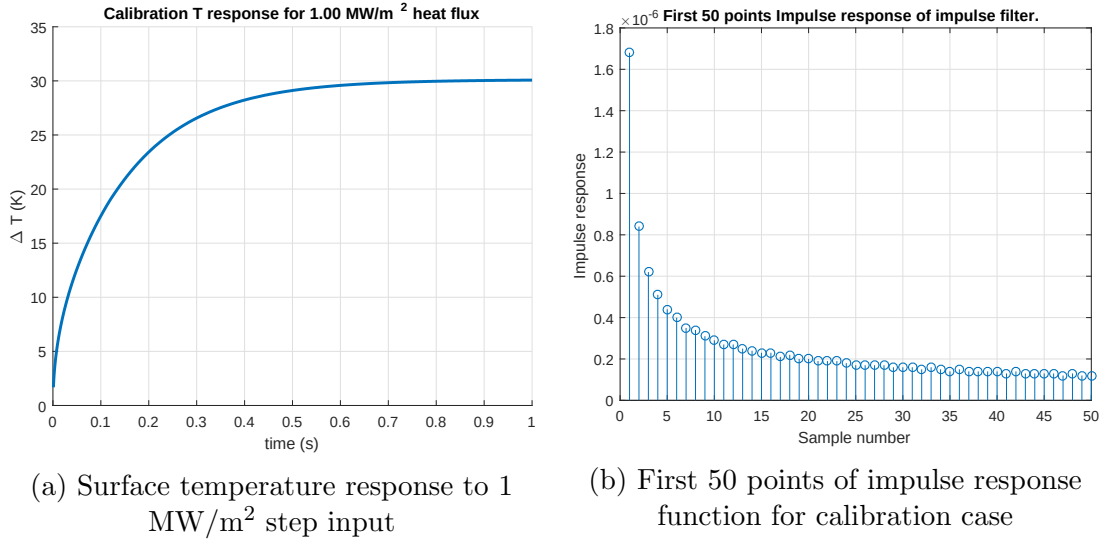
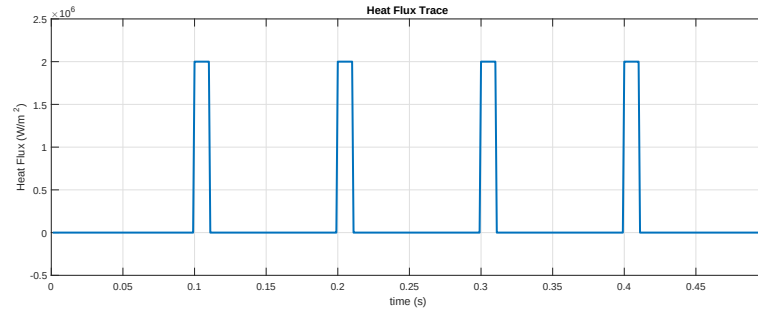
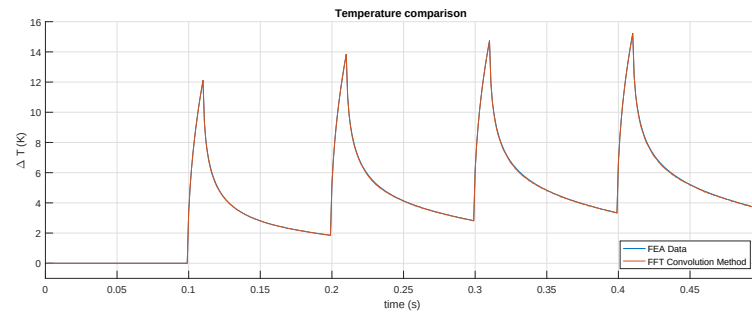


Figure 9.13a shows the surface temperature response on the combustion side to a 1 MW/m^2 heat flux step at $t = 0$. This surface temperature response can then be sampled at the required frequency in order to obtain the impulse response function for the system. The calculated filter was of the same length as the number of samples taken from the surface temperature response.

The first 50 points of the impulse response function can be seen in Figure 9.13b. In order to test the accuracy of the FFT method, the calculated impulse response function (shown in Figure 9.13b) was convolved with three different varying heat flux profiles in order to predict the resultant surface temperature response.

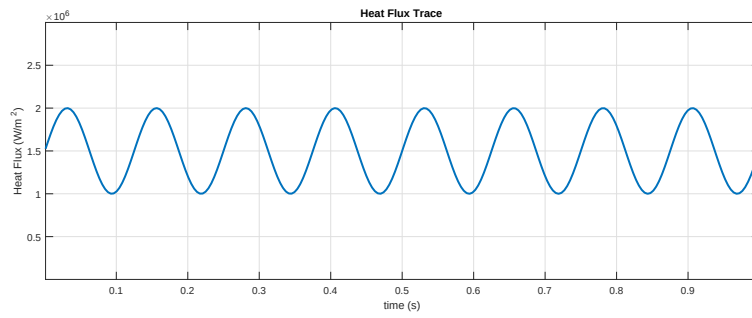


(a)

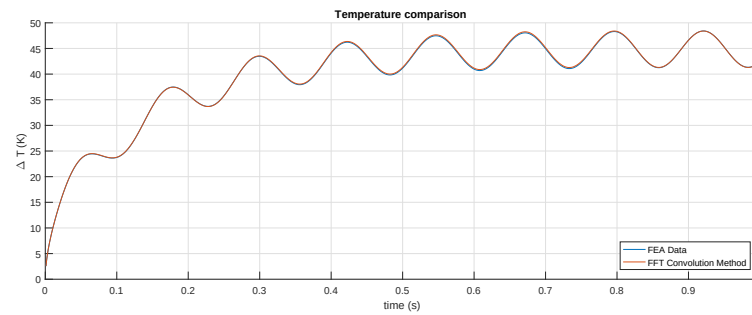


(b)

Figure 9.14: 2MW/m^2 pulse train heat flux input on combustion face



(a)



(b)

Figure 9.15: Sine wave heat flux input on combustion face

Figure 9.14a shows a time varying 2 MW/m^2 pulse train heat flux that was set as a boundary condition to the combustion surface of the finite element model described in Section 9.2.1. The predicted surface temperature profile calculated using the FFT convolution method is shown in Figure 9.14b. This resultant surface temperature profile is directly compared against the surface temperature profile predicted using the 3-D finite element model. As can be seen in Figure 9.14b, the agreement between the two methods is very good.

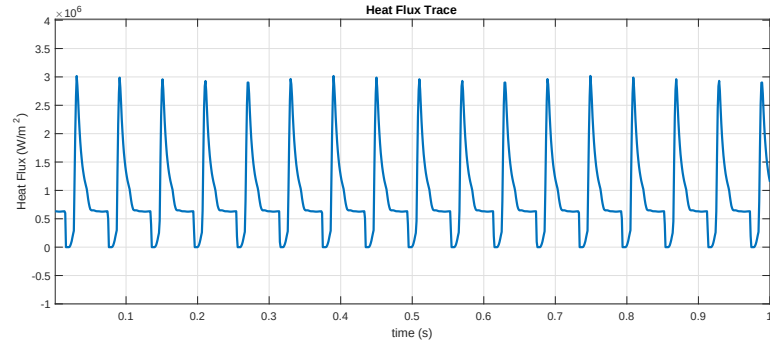
Figure 9.15a shows another time varying heat flux input, set as a boundary condition on the combustion surface of the finite element model. The input to the system was a sine wave with a 1 MW/m^2 amplitude, centered on 1.5 MW/m^2 . As before, the impulse response function was convolved with this heat flux signal in order to predict the surface temperature response of the inner combustion face (shown in Figure 9.15b). This was compared to the surface temperature profile calculated using the 3-D finite element model, also shown in Figure 9.15b. Again, there is little discrepancy between the finite element results and the predicted temperature response using the FFT convolution method.

The last example boundary condition investigated was the heat flux profile shown in Figure 9.16a. This is designed to be a representative cylinder wall heat flux profile for a 4 stroke V6 engine, running at approximately 2000 RPM. The heat flux trace was obtained via a 0-D thermodynamic model of a combustion engine developed in Simulink (adapted from work by Buttsworth et al. [105]).

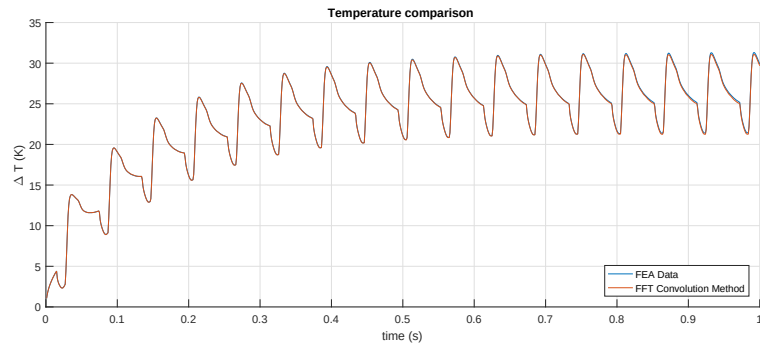
Figure 9.16a shows the heat flux trace obtained from the 0-D model with the cyclic nature of engine cycle evident. Figure 9.16b shows the surface temperature response of the cylinder combustion face as predicted using the FFT method. This is directly compared with the results of the FEA model described in Section 9.2.1. As shown before with the previous examples, the agreement between the FEA and FFT methods is very good with negligible difference between them.

9.2.3 Calculation of coolant effects

A sinusoidal varying coolant temperature (as in Figure 9.8) was again applied to the outer surface of the geometry described in Section 9.2. The effect of this varying



(a)



(b)

Figure 9.16: Example representative combustion heat flux input on combustion face

coolant temperature on the inner surface of the geometry was then calculated using the finite difference method described in Section 8.5 and compared to a 3-D FEA model using the same explicit temperature boundary conditions.

As described in Section 9.1.5.1, an optimisation algorithm was used in order to tune the performance of the finite difference method against the results from a 3-D FEA simulation.

Figure 9.17 shows the temperature of the cylinder inner surface wall with a varying temperature boundary condition placed on the outer surface. As before, the initial settings chosen for the finite difference model showed large discrepancies with the results from the 3-D FEA simulation. Using the gradient descent optimisation method described in Section 9.1.5.1, it was possible to greatly reduce the error observed (taking the 3-D FEA data as the accuracy benchmark) as seen in Figure 9.17. Discrepancies between the tuned model and 3-D FEA data are still evident at start up, but can be seen to reduce as the model behaviour begins to stabilise. Whilst this

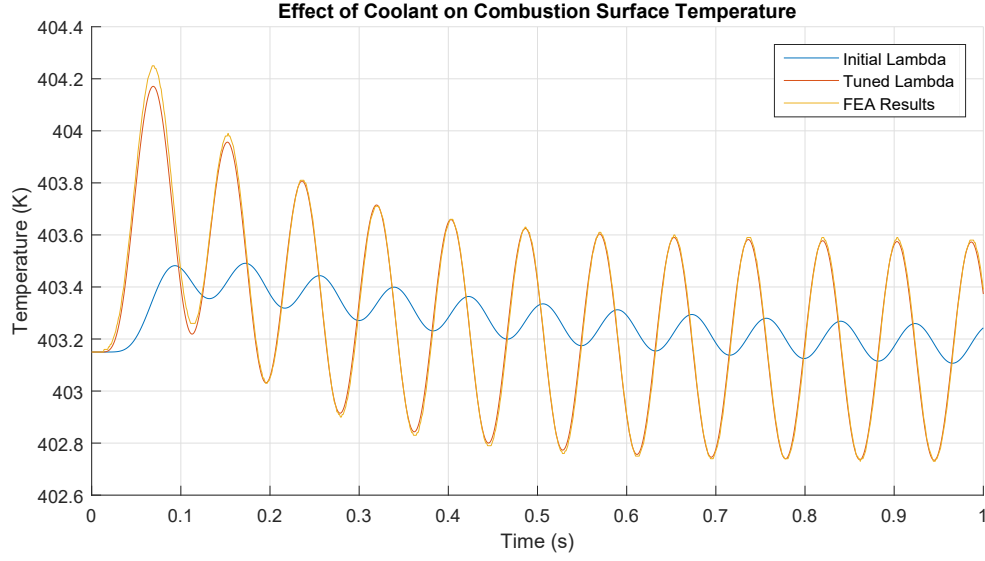


Figure 9.17: Effect of coolant temperature on combustion surface wall for initial and tuned λ values compared with FEA prediction

is not ideal behaviour, the agreement between the finite difference model and the 3-D FEA simulation at steady-state was considered to be more important.

9.2.4 Superposition of results

Figure 9.18 shows the cylinder inner surface wall temperature, subject to the heat flux boundary condition shown in 9.14a with a varying sinusoidal temperature boundary condition (representing the coolant temperature) placed on the cylinder outer surface. The effect of the heat flux boundary condition was calculated using the FFT method, whilst the effect of the temperature boundary condition was solved using the finite difference method. The two individual solutions were then superposed and compared to a 3-D FEA simulation using the same explicit boundary conditions.

As shown in Figure 9.14a, the agreement between the superposed FFT and FD method and the 3-D FEA simulation is very good with minimal discrepancies throughout the simulation period.

Figure 9.19 shows a comparison for the cylinder inner surface wall temperature prediction between superposition method compared with the 3-D FEA model, subject to a sinusoidal heat flux boundary condition (see Figure 9.15a) on the inner face and a varying temperature boundary condition on the outer surface face. The agreement

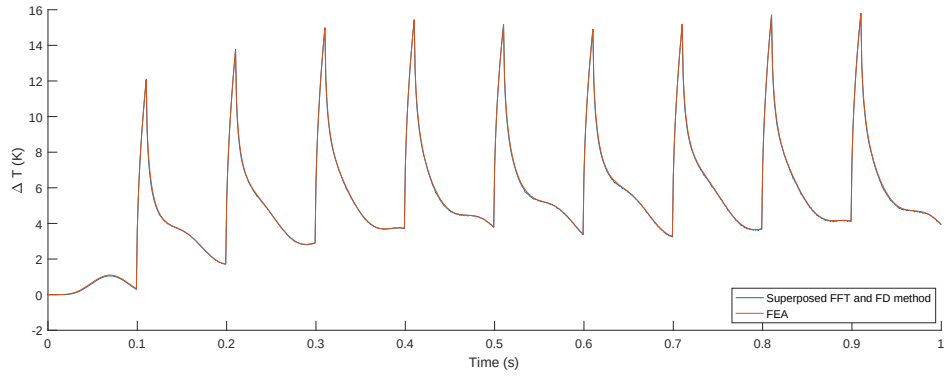


Figure 9.18: Comparison between superposed FFT and FD method and 3-D FEA simulation for pulse train heat flux and varying coolant temperature boundary conditions

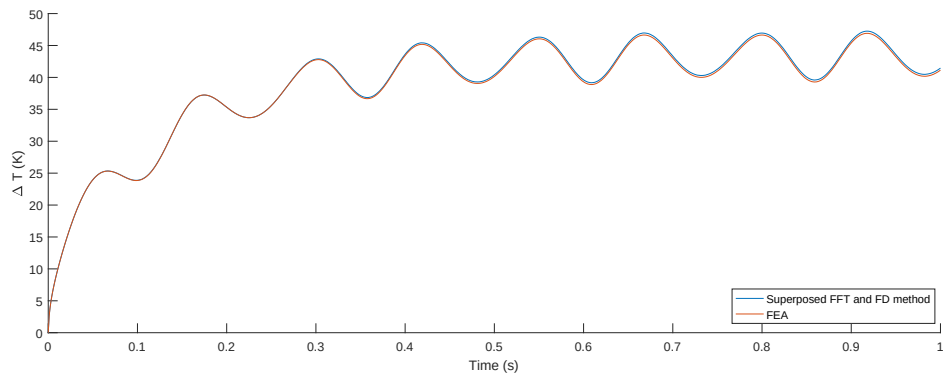


Figure 9.19: Comparison between superposed FFT and FD method and 3-D FEA simulation for sinusoidal heat flux and varying coolant temperature boundary conditions

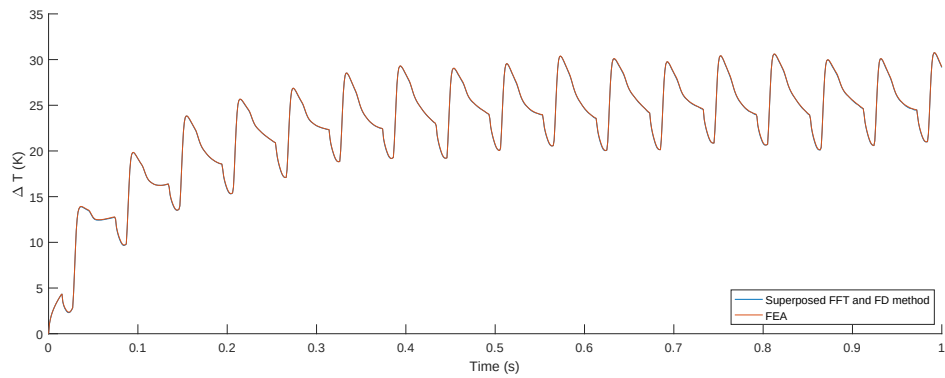


Figure 9.20: Comparison between superposed FFT and FD method and 3-D FEA simulation for combustion process representative heat flux and varying coolant temperature boundary conditions

between the two methods is good, with slight discrepancies in the phase observed (similar to the previous geometry) which is like to be caused by errors in the finite difference model (as a slight discrepancy in phase can be observed in Figure 9.17).

Figure 9.20 shows the superposition and 3-D FEA model comparison for the combustion process representative heat flux boundary condition (see Figure 9.16a) as applied to the cylinder wall inner surface. As before, a varying sinusoidal temperature boundary condition is placed on the cylinder wall outer surface. It is observed that the superposition model matches the 3-D FEA temperature prediction very well.

In Figures 9.18, 9.19, and 9.20, the effect of the varying temperature boundary condition on the outer surface can clearly be seen (compared to the results shown in Section 9.2.2), even though the effect on the surface temperature is relatively minimal compared to the effect of the heat flux boundary condition.

9.2.5 Continuous implementation of cylinder wall model

The examples given in Sections 9.1.6 and 9.2.4 utilise superposition of the FFT and finite difference methods in order to provide an estimate of the cylinder surface temperature in response to a given heat flux and coolant temperature signals. Both examples were post-processed, whereby the heat flux and coolant temperature signals are known to completion beforehand. In reality, the heat flux and coolant temperature signals will be input to the system incrementally during engine operation. Therefore, it was important to consider whether the prediction of the surface temperature can be calculated with the required speed.

In addition to this, the calibration results from the 3-D FEA simulations in Sections 9.1 and 9.2 were the same time period as the inputs to the model. For a real-world engine operation application, it is infeasible to have calibration files that are the same length as the total data that would be input to the system for an engine operation case.

Section 8.6 describes a method for implementing the technique in real-time using a first in, first out (FIFO) stack data structure of heat flux values, whose length is defined by the amount of historical data required to calculate the surface temperature

at the current time point. This allows the technique to be applied to an input with no definable end point, which is necessary for real time operation.

Using the method described in Section 8.6, the same 1 second duration 3-D FEA results (used in Sections 9.2.2 and 9.2.3) were used to tune a continuous implementation of the model described in Section 8.6.

Figure 9.21 shows the heat flux profile applied to the inner face of the cylinder slice geometry described in Section 9.2.1. The coolant temperature shown in Figure 9.22 shows the temperature boundary condition applied to the outer surface of the same geometry. The calibration analyses described in Sections 9.2.2 and 9.2.3 were used to tune the FFT and finite difference models respectively. It should be noted that both the heat flux and coolant temperature profiles shown in Figures 9.21 and 9.22 are longer than the calibration files.

In order to mimic the real-time operation of the system, the heat flux and coolant temperature data were separated into discrete data packets and released sequentially in order to replicate the output of respective sensors in the system. An initial packet length of 10 samples was chosen (which represents 10 milliseconds of data at a sample frequency of 1kHz). Processing of the heat flux and coolant temperature data in each packet to predict the cylinder inner surface temperature was performed when each packet of data was received.

Figure 9.23 shows a comparison between the 3-D FEA data and the real-time implementation of the superposed FFT and finite difference method. It can be seen that the results are very similar to those seen in Sections 9.1.6 and 9.2.4 with the superposition method tracking the predicted behaviour of the system very well, with the difference being less than 0.2 K throughout.

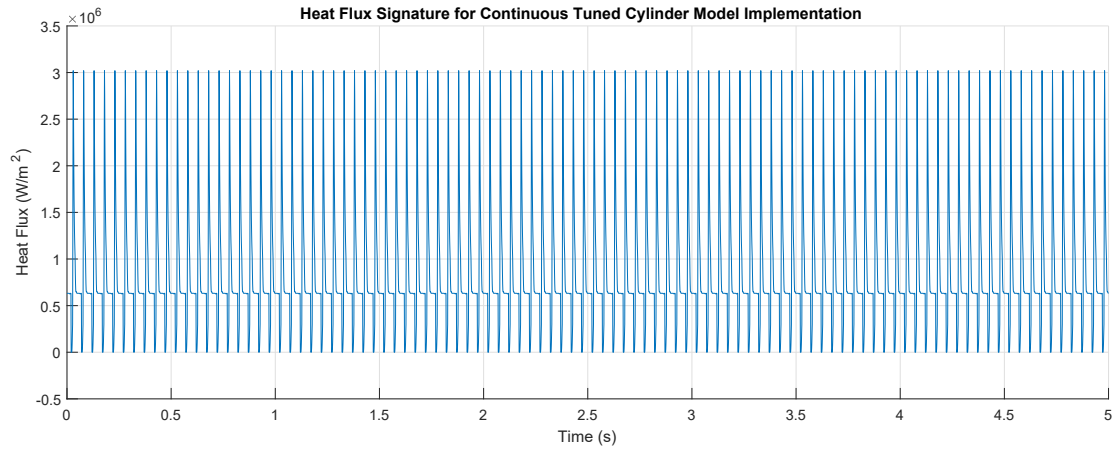


Figure 9.21: Heat flux profile applied to cylinder inner surface

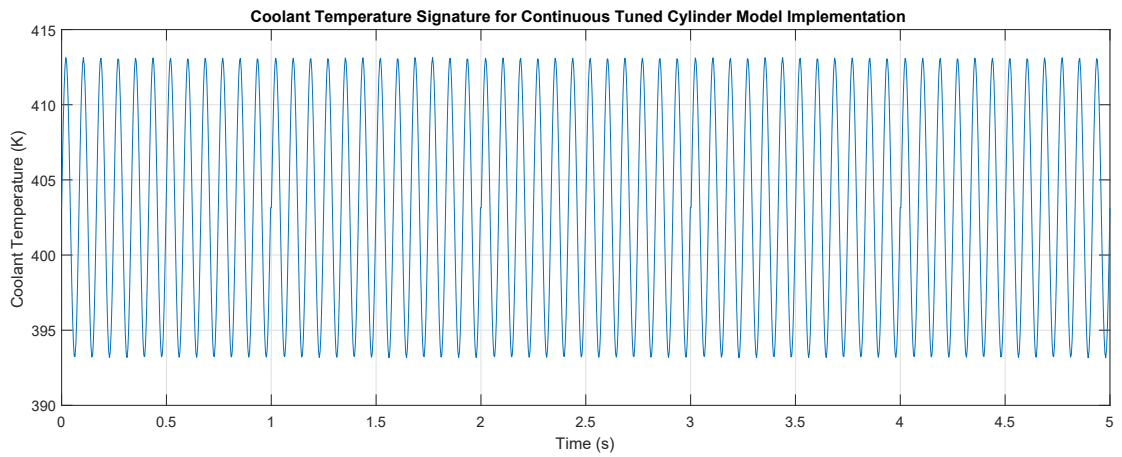


Figure 9.22: Coolant temperature profile applied to cylinder outer surface

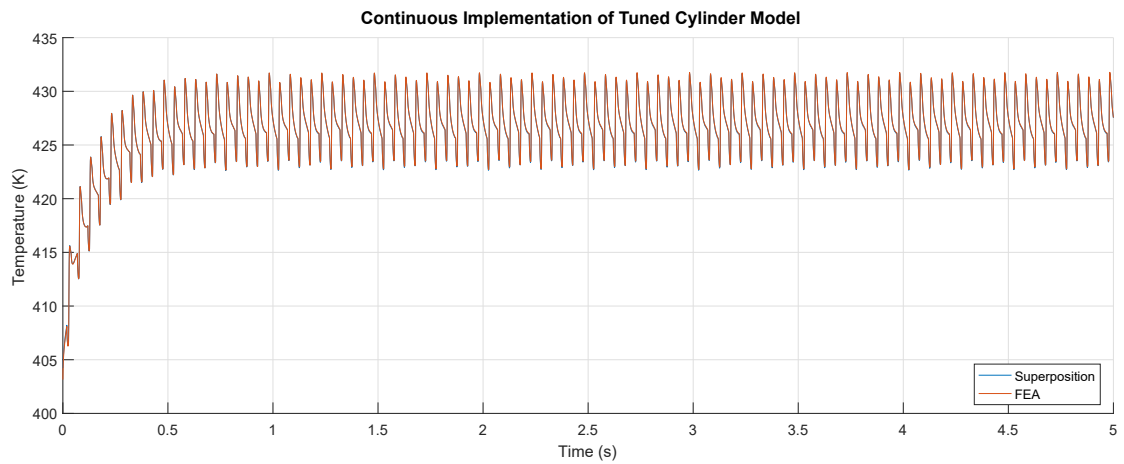


Figure 9.23: Comparison between continuous implementation of superposition method and 3-D FEA simulation of the cylinder inner wall temperature

9.3 Computation Time

9.3.1 Setup and tuning time

The main computational expense of using the superposition method is the cost of acquiring the calibration results files and tuning the model (calculating the impulse response function for the FFT module and multi-parametric optimisation of the finite difference module). However, it should be noted that these are one-off costs, with the relevant simulations calculated before the model is utilised.

Table 9.4: Computation time for FFT and finite difference model tuning

Simulation	Computation Time (seconds)
3-D FEA simulation (1 second)	4881.80
Design of impulse response function	0.0072
Optimisation of finite difference model	191.25

In these examples, acquisition of the calibration result files was performed by running 3-D FEA simulations. The computation time required to run each case is shown in Table 9.4 and was approximately 1 hour and 20 minutes for the geometry shown in Figure 9.12 for each calculation. Alternatively, these results can be acquired through an experimental campaign or through an analytical solution. However, for complex geometries, a 3-D FEA simulation offers the best balance of accuracy and effort.

It would be possible to reduce this time further by decreasing the number of time steps in the 3-D FEA simulations as long as there are enough points to effectively describe the system response, through to steady state behaviour. This is possible as the solution is effectively re-sampled when tuning the model to give the correct behaviour for the required sampling frequency.

Table 9.4 details the time taken to tune the respective FFT and finite difference modules in the system. It is clear that the time taken to derive the the impulse response function is effectively negligible compared to the multi-parametric optimisation study over the material parameters.

The tuning of the respective FFT and finite difference modules is dependent on the system sampling frequency and must be recalculated if this changes. However, the calibration result files (taken from the 3-D FEA simulations) do not need to be re-run, as the results can be re-sampled to match the new system sampling frequency. However, these results are geometry dependent and so if the geometry changes, the calibration simulations must be re-calculated using the computational model and the model re-tuned.

9.3.2 Processing time

Table 9.5: Computation time breakdown per data packet for superposed FFT and finite difference method (80 samples per packet)

Simulation	Computation Time (seconds)
FFT module	0.00015
Finite difference module	0.00009
Combined FFT and finite difference	0.00024

For an engine running at a maximum speed of 15,000 RPM, a single revolution of the crankshaft will take 0.004 seconds, leading to a cycle time of 0.008 seconds (assuming a 4 stroke process). Assuming a sampling frequency of 10 kHz, this would result in approximately 80 samples per cycle. Table 9.5 describes the average time taken to process a packet of data consisting of 80 samples. The mean calculation time of 0.00023972 seconds is approximately an order of magnitude faster than the cycle time, proving that it would be possible to calculate the surface temperature on a cycle-resolved time scale.

The bottleneck to the simulation is the finite difference calculation which requires a double nested loop (as it iterates over each sample in the packet and also over each cell in the model - see Figure 8.5). The simulation was run using a single thread on a Core i5 4570 (Haswell) running at 3.2 GHz with 16 GB of RAM. It would be possible to reduce this computation time further by utilising parallel processing on the inner loop of the finite difference calculation given the data independence of the inner loop.

9.3.3 Comparison to 3-D FEA simulations

Table 9.6: Comparison of computation time for 3-D FEA and superposed simulations

Simulation	Computation Time (seconds)
3-D FEA simulation (1 second)	4881.80
3-D FEA simulation (5 seconds)	24271.91
Superposed FFT and FD simulation (1 second)	0.00763
Superposed FFT and FD simulation (5 seconds)	0.01791

Table 9.6 details the times taken to run various 3-D FEA simulations in ANSYS Workbench. The simulations were completed using a single thread on a Core i7-4770 (Haswell) running at 3.4 GHz with 32 GB of RAM and all of the simulations were run with the same time stepping (0.001 seconds per time step). As expected, for the same geometry, the computation time increases linearly with the number of time steps required.

Table 9.6 describes the time taken to run full simulations using the superposed FFT and finite difference method. It is obvious from the comparison in Table 9.6 that the superposed FFT and finite difference method holds a clear advantage over the 3-D FEA simulations in terms of computation time, with very comparable accuracy.

However, it should be noted that the superposed FFT and finite difference method is only able to return the temperature at a single point which limits its possible applications to either a single point of interest, or environments where the temperature is considered to be homogeneous. It is possible that the same 3-D FEA simulation can be used to calibrate filters for a various points of interest on the same geometry.

9.4 Summary of In-Cylinder Surface Temperature Model Development

Chapters 8 and 9 demonstrated the development of a computationally efficient method of predicting the engine cylinder wall temperature for given heat flux and coolant temperature profiles. The method combined the use of a FFT method (described by

Oldfield [95]) and a finite difference method in order to calculate the respective effects of the heat flux into the cylinder wall and the coolant temperature on the outer face of the engine cylinder.

The FFT model was calibrated using 3-D FEA simulations in order to account for geometrical complexity and a finite back wall temperature. The model was extended to allow continuous predictions of the cylinder wall temperature past the duration of the calibration simulations used to tune the model. The method was also shown to provide good agreement with 3-D FEA simulations and the computational speed of the method was characterised and shown to be compatible with real-time processing during engine operation.

Chapter 10

Summary and Conclusions

This thesis has investigated methods of predicting and measuring various temperature fields in challenging environments. The approach utilised in this body of research combined both experimental and computational techniques in order to understand the heat transfer mechanisms present in both the underhood and combustion engine environments. There are three main outcomes which can be drawn from the work detailed in this thesis.

- A computationally economical simulation tool, based on the linear superposition of solutions to individual boundary condition cases, has been described which allows the prediction of 3-D temperature fields within complex environments.
- The linear superposition model has been validated using a representative experimental test section. A novel method of extracting cross-sectional temperature profiles using low-cost cameras and image processing techniques has been demonstrated.
- Novel heat flux sensors, manufactured using rapid prototyping methods, have been used to measure the heat fluxes within the aggressive environment in an internal combustion engine cylinder under firing conditions.
- A method of rapidly predicting the cylinder wall temperature has been described. The method is based on a simplification of the cylinder wall model and uses the superposition of two individual models in order to solve the system in real-time.

10.1 Predicting temperature fields using linear superposition

The underhood region of a F1 car or consumer vehicle, presents many challenges to the design engineer. The components are subject to a highly complex flow field characterised by re-circulations and high levels of turbulence. The coolant air flow is subject to numerous heat sources so that the air temperature at any location, in many cases, has already been heated by components which are nominally upstream. The experiments reported in this chapter show that it is possible to break the effect of heat sources into individual components by use of the dimensionless temperature influence coefficient introduced in Chapter 3.

10.1.1 Computational Work

A method of predicting 3-D temperature fields in complex geometries has been described and implemented using MATLAB.

The method uses the linear superposition of individual solutions to the temperature field for the separate thermal boundary conditions in the system. As the temperature field solution is governed by a linear, homogeneous partial differential equation, a solution for a complex set of boundary conditions can be found by linearly superposing solutions for each individual boundary condition.

By characterising the influence of each separate thermal boundary conditions on each point in the domain using numerical simulations, the individual solutions can then be scaled and superposed in order to match any combination of thermal boundary conditions to predict the temperature field in the domain. The method has been applied to a quarter scale model of the underhood region of a Formula One race car and has been shown to provide results within 1% agreement of CFD simulations using the same explicit boundary conditions.

This approach has also been extended to factor in changes to the velocity of the working fluid using dimensional analysis techniques. By considering the underhood region as analogous to a ducted heat exchanger, a non-linear Reynolds number correction factor was proposed and shown to reduce the discrepancy between the linear

superposition method and CFD analyses. In order to further improve the agreement between the linear superposition method and CFD analyses, a weighting was then applied to the Reynolds number correction based on patterns observed within the empirical data.

The computational cost of generating temperature field predictions using the linear superposition method, compared to CFD analyses, has been characterised. Utilising the linear superposition method reduces the iterative solution of the momentum and energy equations to basic matrix multiplication operations which can be evaluated using MATLAB. There is a one-off computational cost associated with setting up the linear superposition model, which is the initial characterisation analyses used to calculate the influence of each heat source on each point in the domain. This one-off computational cost is proportional to the number of thermal boundary conditions in the system.

An additional advantage to the fidelity of the initial simulations is carried through to the temperature field predictions made using the linear superposition method. This means that any time saved during the design process can either be used to improve the fidelity of the resulting predictions (such as using more computationally expensive turbulence models or an increased mesh density), to increase the volume of thermal boundary cases investigated, or to shorten the timescale of the design process.

Use of the linear superposition method allows the number of computationally expensive simulations required to be drastically reduced for complex situations where different thermal boundary cases need to be investigated.

10.1.2 Experimental Work

An experimental test facility was used to empirically test elements of the linear superposition model. A quarter scale rear end model of a Formula One race car underhood geometry was created from 20 interlocking slices of Perspex to form an experimental test section. A geometrically simplified set of internal components (comprising engine, turbocharger, exhaust, and gearbox) was manufactured using rapid prototyping and fitted inside the test section.

Two methods of heat addition were used within the domain. The first method utilised electrical heater pads on the internal assembly in order to simulate heat addition from the cam covers of the engine component. The second method used electrical heater meshes at the inlet to the test section in order to generate a step change in gas temperature throughout the test section.

Various sets of instrumentation were fitted inside the experimental test. Orifice meters were fitted at each inlet in order to measure the mass flow through each of the inlets to the test section. Thermocouples were used throughout in order to measure gas temperature at key points within the domain. Thermocouple rakes were also used at the outlet of the test section in order to provide spatial temperature measurements which were then used to evaluate the validity of the linear superposition method.

The heat output from the electrical heater pads was varied in order to change the thermal boundary conditions within the domain with temperature measurements recorded using thermocouple rakes at the outlet to the test section. The results provided experimental validation for the linear superposition method, as the temperatures recorded varied linearly with the output from the heaters. In addition, the results from the individual heater tests were superposed, which showed good agreement with the results from the case with both heaters active simultaneously.

In addition to the aforementioned instrumentation, thermochromic liquid crystals were used to both investigate the cross sectional temperature distribution and to calculate heat transfer coefficients on the exhaust component of the internal assembly. A thermochromic liquid crystal coated nylon mesh was placed in the gas path, which allowed the investigation of the cross-sectional temperature profile.

Both the measured temperature influence coefficients and heat transfer coefficients were then used in order to validate the computational model used to investigate the linear superposition method. Although the use of thermochromic liquid crystals is well documented in literature for transient heat transfer experiments, a novel method of capturing the raw imagery during the experiment using multiple fixed focal length borescopes was utilised. The raw imagery from the separate borescopes was then stitched together (where overlapping areas existed) using image processing techniques

in order to provide more holistic coverage of the part. The work in this section demonstrated an innovative method of introducing low cost borescopes into the underhood region that can produce holistic imagery from individual image sources.

Both the cross sectional temperature distribution and heat transfer coefficients were compared against the computational model described in Chapter 4. The agreement between the active area of the thermochromic liquid crystal coated nylon mesh and the computational simulations was good with the spread and location of high temperature influence coefficient areas being accurately mirrored. The agreement of the heat transfer coefficient was also good, with the same distributions and magnitudes observed in both the experiments and computational simulations. Key differences were observed in areas where the test domain differed, such as discontinuities on the surface of the internal assembly model.

10.2 Measurement of heat fluxes inside an internal combustion engine

The heat flux into the cylinder wall during the engine cycle is a crucial factor in both accurately assessing the life expectancy of key engine components and the efficiency of the engine cycle. Previous experiments that have aimed to measure the heat fluxes during the engine cycle have been based around extensive modifications to the engine under test. The work in this body of research investigated the creation of a heat flux sensor that shares the same mounting interface as a pressure transducer, thus allowing minimal changes to existing test hardware.

In order to meet these challenging geometrical and operation specifications, novel solutions for cable routing and mounting were required. Direct metal laser sintering (DMLS) rapid prototyping was used to create the parts for the sensor, which allowed a novel method of cable routing to be trialled.

The two layer gauge method (using an electrically insulating coating on a base metal substrate) was shown to be ineffective due to the geometrical constraints imposed by the chosen form factor. However, application of platinum thin film gauges to both coatings was shown to be possible when these geometrically constraints were removed.

The heat flux sensor was based around thin film gauge technology, which uses a resistive platinum sensor to measure temperature via changes in the sensor resistance. The thin film gauge was deposited on a ceramic substrate which can be considered semi-infinite. The heat flux can then be calculated from the surface temperature measurements through solving the 1-D Fourier equation using the FFT method outlined by Oldfield [95].

The heat flux sensor was tested in a single cylinder, 4-stroke petrol engine. The cylinder head was modified to include two identical mounts in order to measure both pressure and heat flux. The resulting heat flux data was then compared with two empirical heat transfer models (the Woschni [5] and Hohenberg [6] models) that are typically used for combustion engine applications.

It was seen that the heat flux measured did not fully agree with the theoretical models. The Hohenberg model was seen to predict the peak heat flux value within 5%, while the Woschni model tended to grossly over predict the peak heat flux value for firing conditions.

An amorphous diamond coating was trialled in order to increase the robustness of the sensor. It was found to increase the sensor's resistance to abrasion during engine operation but the effect was limited due to restrictions on the coating thickness that could be applied. A coated sensor was used in the single cylinder test facility and was found to give similar results to an uncoated sample. This was expected due to the high thermal conductivity of the amorphous diamond coating. Given the improvements in sensor robustness and minimal impact on the sensing circuit performance, the use of an amorphous diamond coating could allow a robust thin film gauge to be created for more suitable geometries.

10.3 Predicting cylinder wall temperature using fast Fourier transform method and finite difference method

A method for predicting the cylinder wall temperature in real-time has been described and implemented in MATLAB.

The method simplified the system to a conduction heat transfer problem through a small slice of the cylinder wall, with the thermal boundary conditions on each face set by the heat flux from the combustion gases and the coolant temperature respectively. The effect of each thermal boundary was separated into two individual models which are independently solved and superposed.

The method was shown to give excellent agreement with 3-D finite element analysis (FEA) simulations. In addition, the method has been implemented both for post-processing and for real-time applications. The computation cost of the method is incredibly important for real-time applications, and as such, it was shown that the cost of operating over 80 data points can be completed in under 1ms. For post-processing applications, the method was approximately six orders of magnitude quicker than a 3-D FEA simulation.

10.3.1 Use of fast Fourier transform method to predict temperature

The use of fast Fourier transforms in order to calculate heat fluxes from temperature is well documented in literature for experimental post processing applications [95]. The same method can be applied to perform the reverse transformation (calculating resultant temperature from heat flux). This was the approach used to calculate the effect of the combustion cycle heat flux on the cylinder wall temperature.

The use of fast Fourier transforms for calculating heat fluxes from temperature has traditionally been used to post-process experimental results. In this application, a time marching method was implemented, using a first-in, first-out stack data structure in order to implement the method for real-time applications.

In previous uses of the fast Fourier transform method, the impulse response function was calibrated using an analytical solution to the 1-D Fourier equation for a step change in heat flux. This was enabled by the assumption of a semi-infinite boundary condition. In this application, the impulse response function was calculated using a 3-D FEA solution which allows a finite temperature boundary condition on the opposite face. The use of a 3-D FEA solution for the calculation of the impulse response

function also allows for geometrical complexity (such as curvature) and multi-layer systems.

The predicted heat release profile from the combustion cycle (which can be calculated using measured parameters, such as pressure) can then be filtered using the impulse response function in order to calculate its effect on the cylinder wall temperature.

10.3.2 Use of finite differencing to calculate coolant effect

The effect of the coolant temperature is a key factor in managing the thermal loads on the engine components throughout the combustion cycle, and as such, has an effect on the cylinder wall temperature. This effect was captured in the model by a finite difference method.

Similarly to the effect of the combustion heat flux, the effect of the coolant temperature was calculated by solving the 1-D heat conduction equation through the cylinder wall. The heat transfer coefficient between the coolant and rear face of the cylinder wall was deemed to be sufficiently high, such that a temperature boundary condition (equal to the coolant temperature) was placed on the rear cylinder face.

The finite difference model can be tuned by varying the time step parameter for the model. A 3-D FEA model is used to provide a calibration solution, which the finite difference model was then tuned against. A cost function was defined using the difference between the finite difference model and the FEA simulation, which is then minimised using a gradient descent optimisation algorithm.

10.4 Recommendations for future work

The work in this thesis can broadly be separated into three categories. As such, recommendations for future work can also be separated into the same three categories.

10.4.1 Linear superposition method

The linear superposition method has been tested and implemented for steady state simulations. As the method is placed as an computationally economical alternative to full computational simulations, it is highly possible that end-users may want to

use any time saved in order to investigate transient simulations using the linear superposition method. For underhood applications, this is particularly relevant for cold start and thermal soak conditions.

For varying thermal boundary conditions, the application of the linear superposition method to transient simulations should be fairly straightforward as transience can be approximated using steady state simulations at each time step with linked boundary conditions. However, for varying inlet velocities, this creates changes in the flow field which will not necessarily be captured by the linear superposition method. It is likely that time lag corrections would need to be applied across the domain, linked to the influence of the inlet velocity on the velocity at each point in the domain (which could be considered analogous to the temperature influence matrix described in Section 3.4). Extending the method to allow the investigation of transient boundary conditions would broadly extend the appeal of using the linear superposition method, particularly given the computational expense associated with running transient simulations.

The second piece of work that would help drive the use of the linear superposition method would be to apply the process to other test geometries. In this body of work, the application of the linear superposition method has been limited to two geometries. By expanding the amount of test cases, this would strengthen the case to adopt the linear superposition method in favour of traditional wholly CFD based design methodologies.

Linked to this would be further experimental validation of the method, particularly for the velocity scaling techniques. This would help to characterise the assumptions made in the velocity scaling model, and also help to further refine the proposed correction which aims to account for the effect of Reynolds number. An additional smaller body of work would include the effect of thermal inertia in the system using conjugate CFD or similar. This is coupled to a desire to investigate transient simulations to allow for thermal soak investigations.

Experimental validation is vastly important in order to reinforce the value of using the linear superposition method. The use of thermochromic liquid crystals in order to calculate cross sectional temperature profiles was limited by the use of a narrow

band crystal as it was only possible to extract useful information when the mesh was within the activation band of the thermochromic liquid crystal. This limited the combination of thermal and inlet boundary conditions that could be investigated. This could be improved through the use of a thermochromic liquid crystal with a wider activation band.

10.4.2 Internal combustion engine heat flux measurement

A heat transfer sensor was designed to measure the heat flux inside an internal combustion engine during operation.

Further work in this body of research should include testing the sensor across a variety of different engines. However, improving the robustness of the thin film gauge should first be prioritised.

The use of an alumina-zirconia coating in a multi-layer gauge system has the potential to be feasible systems for high temperature and high pressure applications (such as instrumentation of an internal combustion engine) due to its material properties. If the geometrical constraints are relaxed, then use of an insulating coating (such as vitreous enamel or alumina-zirconia) could potentially be used for sensor applications in aggressive environments. The alumina-zirconia coating is purported to be stable at temperatures up to 1200°C. For high heat flux applications, it may also be necessary to investigate the use of active cooling strategies to manage the temperature of the coating.

As stated in this body of research, it was observed that the platinum thin film gauge was used until the sensing circuit became open circuit. An amorphous diamond coating was applied over the thin film gauge. However, the experiments showed that the adhesion of the diamond coating was not enough to prevent abrasion of the coating and the gauge. The use of an amorphous diamond coating shows great promise if the thickness of the coating could be improved, which would necessitate relaxing the geometrical constraints. If the amorphous diamond coating method is pursued, it would be recommended that an effective way of accurately determining the thermal product and thickness of the amorphous diamond coating be developed in order to improve the accuracy of the heat flux calculations.

One possible method of prolonging the life of the sensor would be to use a temporary insertion technique that would be able to expose the gauge for a limited number of cycles. This would still yield useful information given the typical speed of a combustion engine, meaning that even a one second insertion would still be able to capture multiple cycles. However, this would require significant modifications to the experimental rig which was an factor that the design aimed to avoid.

It should be noted that the finite element analysis showed that the temperature of the ceramic substrate under the observed conditions in the single cylinder test bench was not considered to be a factor in the thin film gauge failure. However, under increased thermal loading, it is possible that the temperature of the ceramic substrate might then become a factor in the sensor life expectancy.

10.4.3 Cylinder wall temperature prediction using a tuned thermal model

The tuned cylinder wall model has been shown to provide good agreement with 3-D FEA simulations. The next step in driving adoption of this technique would be to implement it on application specific hardware and for experimental data.

This would likely be deployment on an embedded solution (such as an engine control unit) which would allow the computation speed of the system to be assessed on relevant hardware. An additional piece of work would be to use experimental data to tune the model. This could be run in parallel with experimental validation of the model.

As computation speed is a major factor in the feasibility of implementing the method, optimisation of the code would also be recommended, particularly the utilisation of parallel processing methods. This would likely yield a significant improvement in speed, given the data independence within the system and would also follow the industry trend of adopting many-core architectures for engine control units.

Bibliography

- [1] S. Collins, “The Parisian Powerhouse,” *Racecar Engineering*, pp. 46–48, 2012.
- [2] United States Department of Energy, “DOE Co-Funds 12 Projects to Increase Engine Efficiency,” 2005.
- [3] Federation Internationale de L’Automobile, “Formula One World Championship Technical Regulations 2017,” 2017.
- [4] Federation Internationale de L’Automobile, “Formula One World Championship Sporting Regulations 2017,” 2017.
- [5] G. Woschni, “A Universally Applicable Equation for the Instantaneous Heat Transfer Coefficient in the Internal Combustion Engine,” *SAE Technical Paper 670931*, 1967.
- [6] G. F. Hohenberg, “Advanced Approaches for Heat Transfer Calculations,” *SAE Technical Paper 790825*, 1979.
- [7] C. Lim, P. T. Ireland, and N. Collett, “Analysing Underhood Temperature Fields Using Linear Superposition,” in *SAE Thermal Management Systems Symposium*, (Detroit, Michigan), 2016.
- [8] C. Lim, P. T. Ireland, and N. Collett, “Analysis of Underhood Temperature Fields using Linear Superposition,” *SAE Technical Paper 2017-01-0138*, 2017.
- [9] R. H. Barnard, “Theoretical and experimental investigation of the aerodynamic drag due to automotive cooling systems,” *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, vol. 214, no. 8, pp. 919–927, 2000.

- [10] M. Khaled, H. Fabien, and P. Hassan, "Towards the control of car underhood thermal conditions," *Applied Thermal Engineering*, vol. 31, no. 5, pp. 902–910, 2011.
- [11] F. Fortunato, F. Damiano, L. D. Matteo, and P. Oliva, "Underhood Cooling Simulation for Development of New Vehicles," *SAE Technical Paper 2005-01-2046*, 2005.
- [12] M. Khaled, F. Harambat, and H. Peerhossaini, "Underhood thermal management: Temperature and heat flux measurements and physical analysis," *Applied Thermal Engineering*, vol. 30, no. 6, pp. 590–598, 2009.
- [13] R. Pamoti and C. Raval, "Underhood Flow Analysis of a LCV to Improve Cooling System Performance for Better Fuel Economy," in *Altair Technology Conference 2013*, (India), 2013.
- [14] M. Khaled, F. Harambat, and H. Peerhossaini, "Temperature and Heat Flux Behavior of Complex Flows in Car Underhood Compartment," *Heat Transfer Engineering*, vol. 31, no. 13, pp. 1057–1067, 2010.
- [15] C. Brodbeck, "Automotive Thermal Management for Full Vehicles Motivation and Problem Description," tech. rep., Fraunhofer SCAI Internal Magazine, 2015.
- [16] I. Bayraktar, "Computational simulation methods for vehicle thermal management," *Applied Thermal Engineering*, vol. 36, pp. 325–329, 2012.
- [17] S. Mao, Z. Feng, and E. E. Michaelides, "Off-highway heavy-duty truck underhood thermal analysis," *Applied Thermal Engineering*, vol. 30, pp. 1726–1733, 2010.
- [18] E. Ljungskog and U. Nilsson, *CFD for Underhood Modeling*. Master's thesis, Chalmers University of Technology, 2014.
- [19] A. Alajbegovic, R. Sengupta, and W. Jansen, "Cooling Airflow Simulation for Passenger Cars using Detailed Underhood Geometry," *SAE Technical Paper 2006-01-3478*, 2006.

- [20] R. Stout, “Linear Superposition Speeds Thermal Modelling,” tech. rep., Power Electronics, 2007.
- [21] P. Gauché, “A design approach to thermal characterization of forced convection systems using superposition in CFD,” in *SPIE proceedings series*, pp. 530–535, Society of Photo-Optical Instrumentation Engineers, 2000.
- [22] P. Gauché and T. Road, “Using FLOTHERM and the Command Center to Exploit the Principle of Superposition,” in *9th International FLOTHERM User Conference*, 2013.
- [23] J. W. Sofia, “Electrical Thermal Resistance Measurements for Hybrids and Multi-Chip Packages,” tech. rep., Analysis Technology Inc., 1990.
- [24] P. Gauché and W. Wei, “Linearized superposition using CFD for thermal and power characterization of electronic equipment with significant thermal radiation and natural convection,” in *IMAPS 2002: 35th International Symposium on Microelectronics*, (Denver, CO), IMAPS, 2002.
- [25] B. Zahn, “Steady state thermal characterization of multiple output devices using linear superposition theory and a non-linear matrix multiplier,” in *14th Annual IEEE Semiconductor Thermal Measurement and Management Symposium*, pp. 39–46, IEEE, 1998.
- [26] G. Borman and K. Nishiwaki, “Internal-combustion engine heat transfer,” *Progress in Energy and Combustion Science*, vol. 13, no. 1, pp. 1–46, 1987.
- [27] R. Stone, *Introduction to Internal Combustion Engines*. SAE International, 2nd ed., 1985.
- [28] C. A. Finol and K. Robinson, “Thermal modelling of modern engines: a review of empirical correlations to estimate the in-cylinder heat transfer coefficient,” *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, vol. 220, no. 12, pp. 1765–1781, 2006.
- [29] F. P. Incropera and D. P. DeWitt, *Fundamentals of Heat and Mass Transfer*. Wiley, 3rd ed., 1990.

- [30] W. J. D. Annand, "Heat transfer in the cylinders of reciprocating internal combustion engines," *Proceedings of the Institution of Mechanical Engineers*, vol. 177, no. 36, pp. 973–996, 1963.
- [31] N. Watson and M. S. Janota, *Turbocharging the Internal Combustion Engine*. Palgrave Macmillan, 1st ed., 1982.
- [32] W. J. Annand and T. H. Ma, "Instantaneous heat transfer rates to the cylinder head surface of a small compression-ignition engine," *Proceedings of the Institution of Mechanical Engineers*, vol. 185, no. 72, pp. 976–987, 1971.
- [33] J. Chang, O. Güralp, Z. Filipi, D. N. Assanis, T.-W. Kuo, P. Najt, and R. Rask, "New Heat Transfer Correlation for an HCCI Engine Derived from Measurements of Instantaneous Surface Heat Flux," *SAE Technical Paper 2004-01-2996*, 2004.
- [34] A. Wimmer, R. Pivec, and T. Sams, "Heat Transfer to the Combustion Chamber and Port Walls of IC Engines - Measurement and Prediction," *SAE Technical Paper 2000-01-0568*, 2000.
- [35] D. L. Schultz and T. V. Jones, "Heat-Transfer Measurements in Short-Duration Hypersonic Facilities," tech. rep., Advisory Group for Aerospace Research and Development, Paris, 1973.
- [36] J. A. Gatowski, M. K. Smith, and A. C. Alkidas, "An experimental investigation of surface thermometry and heat flux," *Experimental Thermal and Fluid Science*, vol. 2, no. 3, pp. 280–292, 1989.
- [37] R. Vidal, "Model instrumentation techniques for heat transfer and force measurements in a hypersonic shock tunnel," tech. rep., Cornell Aeronautical Laboratory, Buffalo N.Y., 1956.
- [38] T. V. Jones, "Gas Turbine Studies at Oxford 1969-1987," in *ASME 1988 International Gas Turbine and Aeroengine Congress and Exposition*, (Amsterdam), p. V004T09A017, ASME, 1988.
- [39] T. S. Wilson, P. J. Bryanston-Cross, K. S. Chana, P. Dunkley, T. V. Jones, and P. Hannah, "High Bandwidth Heat Transfer and Optical Measurements in

- an Instrumented Spark Ignition Internal Combustion Engine,” *SAE Technical Paper 2002-01-0747*, 2002.
- [40] J. Demuynck, S. Pauwels, S. Verhelst, M. De Paepe, and S. Roger, “Experimental research on the heat transfer inside a hydrogen combustion engine: evaluation and construction of measurement methods,” *FISITA World Automotive Congress 2008*, 2008.
 - [41] D. Bendersky, “A special thermocouple for measuring transient temperatures,” *Mechanical Engineering*, vol. 75, no. 2, pp. 117–121, 1953.
 - [42] K. Sihling and G. Woschni, “Experimental investigation of the instantaneous heat transfer in the cylinder of a high speed diesel engine,” *SAE Technical Paper 790833*, 1979.
 - [43] V. D. Overbye, J. E. Bennethum, O. A. Uyehara, and P. S. Myers, “Unsteady Heat Transfer in Engines,” *SAE Technical Paper 610041*, 1961.
 - [44] D. R. Buttsworth, “Assessment of effective thermal product of surface junction thermocouples on millisecond and microsecond time scales,” *Experimental Thermal and Fluid Science*, vol. 25, no. 6, pp. 409–420, 2001.
 - [45] A. C. Alkidas, “Heat Transfer Characteristics of a Spark-Ignition Engine,” *Journal of Heat Transfer*, vol. 102, no. 2, p. 189, 1980.
 - [46] C. R. Stone, E. P. Lim, P. Ewart, G. Lloyd, and R. B. Williams, “Temperature and Heat Flux Measurements in a Spark Ignition Engine,” *SAE Technical Paper 2000-01-1214*, 2000.
 - [47] X. Wang, R. Stone, R. Stevens, Y. Arita, and D. Buttsworth, “Finite Element Analysis of Eroding Type Surface Thermocouple with Application to Engine Heat Flux Measurement,” *SAE Technical Paper 2006-01-1045*, 2006.
 - [48] X. Wang, P. Price, R. Stone, and D. Richardson, “Burn Rate and Instantaneous Heat Flux Study of Iso-octane, Toluene and Gasoline in a Spray-Guided Direct-Injection Spark-Ignition Engine,” *SAE Technical Paper 2008-01-0469*, 2008.

- [49] D. R. Buttsworth, R. Stevens, and C. R. Stone, "Eroding ribbon thermocouples: impulse response and transient heat flux analysis," *Measurement Science and Technology*, vol. 16, no. 7, pp. 1487–1494, 2005.
- [50] C. Rakopoulos and G. Mavropoulos, "Experimental instantaneous heat fluxes in the cylinder head and exhaust manifold of an air-cooled diesel engine," *Energy Conversion and Management*, vol. 41, no. 12, pp. 1265–1281, 2000.
- [51] D. R. Buttsworth, "Transient response of an erodable heat flux gauge using finite element analysis," *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, vol. 216, no. 8, pp. 701–706, 2002.
- [52] X. Wang, P. Price, C. R. Stone, and D. Richardson, "Heat release and heat flux in a spray-guided direct-injection gasoline engine," *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, vol. 221, no. 11, pp. 1441–1452, 2007.
- [53] A. Mohammadi and M. Yaghoubi, "Estimation of instantaneous local heat transfer coefficient in spark-ignition engines," *International Journal of Thermal Sciences*, vol. 49, no. 7, pp. 1309–1317, 2010.
- [54] Y. Liu and R. Reitz, "Modeling of heat conduction within chamber walls for multidimensional internal combustion engine simulations," *International Journal of Heat and Mass Transfer*, vol. 41, no. 6, pp. 859–869, 1998.
- [55] J. Xin, S. Shih, E. Itano, and Y. Maeda, "Integration of 3D Combustion Simulations and Conjugate Heat Transfer Analysis to Quantitatively Evaluate Component Temperatures," *SAE Technical Paper 2003-01-3128*, 2003.
- [56] Y. Li and S.-C. Kong, "Coupling conjugate heat transfer with in-cylinder combustion modeling for engine simulation," *International Journal of Heat and Mass Transfer*, vol. 54, no. 11, pp. 2467–2478, 2011.
- [57] Z. Hu, "Non-Linear Instabilities of Combustion Processes and Cycle-to-Cycle Variations in Spark-Ignition Engines," in *SAE Technical Paper 961197*, SAE International, 1996.

- [58] F. E. Nelson, *Measurement and analysis of spark ignition engine pressure data to determine heat release profiles*. PhD thesis, Massachusetts Institute of Technology, 1984.
- [59] D. Moro, N. Cavina, and F. Ponti, “In-Cylinder Pressure Reconstruction Based on Instantaneous Engine Speed Signal,” *Journal of Engineering for Gas Turbines and Power*, vol. 124, no. 1, pp. 220–225, 2001.
- [60] P. Zeng and D. N. Assanis, “Reconstructing Cylinder Pressure of a Spark-Ignition Engine for Heat Transfer and Heat Release Analyses,” in *ASME 2004 Internal Combustion Engine Division Fall Technical Conference*, no. 37467, pp. 43–52, 2004.
- [61] J. A. Gatowski, E. N. Balles, K. M. Chun, F. E. Nelson, J. A. Ekchian, and J. B. Heywood, “Heat Release Analysis of Engine Pressure Data,” in *SAE Technical Paper 841359*, SAE International, 1984.
- [62] M. F. J. Brunt, H. Rai, and A. L. Emtage, “The Calculation of Heat Release Energy from Engine Cylinder Pressure Data,” in *SAE Technical Paper 981052*, SAE International, 1998.
- [63] L. Eriksson, *Spark Advance Modeling and Control*. PhD thesis, Linköping University, 1999.
- [64] I. Glaser and J. D. Powell, “Optimal Closed-Loop Spark Control of an Automotive Engine,” *SAE Technical Paper 810058*, 1981.
- [65] L. Eriksson, “Spark advance for optimal efficiency,” *SAE Technical Paper 1999-01-0548*, 1999.
- [66] M. Hubbard, P. D. Dobson, and J. D. Powell, “Closed Loop Control of Spark Advance Using a Cylinder Pressure Sensor,” *Journal of Dynamic Systems, Measurement, and Control*, vol. 98, no. 4, pp. 414–420, 1976.
- [67] J. B. Heywood, *Internal Combustion Engine Fundamentals*. McGraw-Hill, 1st ed., 1988.
- [68] J.-H. Lu, D. Ezekoye, A. Iiyama, R. Greif, and R. F. Sawyer, “Effect of Knock on Time-Resolved Engine Heat Transfer,” *SAE Technical Paper 890158*, 1989.

- [69] E. Corti and C. Forte, “A Statistical Approach to Spark Advance Mapping,” *Journal of Engineering for Gas Turbines and Power*, vol. 132, no. 8, p. 082803, 2010.
- [70] J. Demuynck, M. De Paepe, L. Sileghem, J. Vancoillie, S. Verhelst, and K. Chana, “Applying Design of Experiments to Determine the Effect of Gas Properties on In-Cylinder Heat Flux in a Motored SI Engine,” *SAE International Journal of Engines*, vol. 5, no. 3, pp. 2012–01–1209, 2012.
- [71] E. Kreyszig, *Advanced Engineering Mathematics*. John Wiley & Sons, 10th ed., 2010.
- [72] M. J. Holland and T. F. Thake, “Rotor blade cooling in high pressure turbines,” *Journal of Aircraft*, vol. 17, no. 6, pp. 412–418, 1980.
- [73] F. P. Incropera, D. P. DeWitt, T. L. Bergman, and A. S. Lavine, *Fundamentals of Heat and Mass Transfer*. John Wiley & Sons, 2006.
- [74] R. Maffulli and L. He, “Wall Temperature Effects on Heat Transfer Coefficient,” in *ASME Turbo Expo 2013: Turbine Technical Conference and Exposition*, no. 55164, p. V03CT14A003, 2013.
- [75] P. Forsyth, M. McGilvray, and D. R. H. Gillespie, “Secondary flow and heat transfer coefficient distributions in the developing flow region of ribbed turbine blade cooling passages,” *Experiments in Fluids*, vol. 58, no. 1, p. 5, 2016.
- [76] M. McGilvray, D. Gillespie, and J. Ryley, “Investigation of Wrapping Ribs Onto Smooth Walls for Mid-Chord Internal Cooling Passages,” in *ASME Turbo Expo 2014: Turbine Technical Conference and Exposition*, no. 45714, p. V05AT12A045, 2014.
- [77] R. Pearce, P. Ireland, L. He, M. McGilvray, and E. Romero, “Computational and Experimental Study of Heat Transfer in Rotating Ribbed Radial Turbine Cooling Passages,” in *ASME Turbo Expo 2015: Turbine Technical Conference and Exposition*, no. 56710, p. V05AT11A026, 2015.

- [78] P. T. Ireland, A. J. Neely, D. R. Gillespie, and A. J. Robertson, “Turbulent heat transfer measurements using liquid crystals,” *International Journal of Heat and Fluid Flow*, vol. 20, no. 4, pp. 355–367, 1999.
- [79] D. R. Gillespie, P. T. Ireland, and Z. Wang, “British Patent Application: Heating Element (PCT/GB1996/002017),” 1995.
- [80] Z. Wang, P. T. Ireland, T. V. Jones, and R. Davenport, “A color image processing system for transient liquid crystal heat transfer experiments,” *ASME Journal of Turbomachinery*, vol. 118, no. 3, 1996.
- [81] Z. Wang, P. T. Ireland, and S. T. Kohler, “Gas temperature measurement in internal cooling passages,” in *ASME 1996 International Gas Turbine and Aero-engine Congress and Exhibition*, American Society of Mechanical Engineers, 1996.
- [82] C. Camci, K. Kim, and S. A. Hippensteele, “A New Hue Capturing Technique for the Quantitative Interpretation of Liquid Crystal Images Used in Convective Heat Transfer Studies,” *Journal of Turbomachinery*, vol. 114, no. 4, pp. 765–775, 1992.
- [83] R. Poser, J. von Wolfersdorf, and E. Lutum, “Advanced evaluation of transient heat transfer experiments using thermochromic liquid crystals,” *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 221, no. 6, pp. 793–801, 2007.
- [84] D. Capel, “Image Mosaicing,” in *Image Mosaicing and Super-resolution* (D. Capel, ed.), pp. 47–79, London: Springer London, 2004.
- [85] A. Murray, *Visualisation of the 3D Temperature Field within the Engine Bay of a Formula One Vehicle*. 4YP Report, University of Oxford, 2014.
- [86] R. J. Clifford, T. V. Jones, and S. T. Dunne, “Techniques for obtaining detailed heat transfer coefficient measurements within gas turbine blade and vane cooling passages,” in *ASME 1983 International Gas Turbine Conference and Exhibit*, American Society of Mechanical Engineers, 1983.

- [87] A. R. A. Talib, A. J. Neely, P. T. Ireland, and A. J. Mullender, “A Novel Liquid Crystal Image Processing Technique Using Multiple Gas Temperature Steps to Determine Heat Transfer Coefficient Distribution and Adiabatic Wall Temperature,” *Journal of Turbomachinery*, vol. 126, no. 4, pp. 587–596, 2004.
- [88] M. McGilvray and D. R. Gillespie, “Transient Heat Transfer Analysis Code for Liquid Crystal Experiments: User Manual,” tech. rep., University of Oxford, 2011.
- [89] T. V. Jones, “Gas Turbine Studies at Oxford 1969–1987,” in *Volume 4: Heat Transfer; Electric Power; Industrial and Cogeneration*, p. V004T09A017, ASME, 1988.
- [90] E. Piccini, S. M. Guo, and T. V. Jones, “The development of a new direct-heat-flux gauge for heat-transfer facilities,” *Measurement Science and Technology*, vol. 11, no. 4, pp. 342–349, 2000.
- [91] J. E. Doorly and M. L. G. Oldfield, “New Heat Transfer Gages for Use on Multilayered Substrates,” *Journal of Turbomachinery*, vol. 108, no. 1, p. 153, 1986.
- [92] R. W. Ainsworth, J. L. Allen, M. R. D. Davies, J. E. Doorly, C. J. P. Forth, M. A. Hilditch, M. L. G. Oldfield, and A. G. Sheard, “Developments in Instrumentation and Processing for Transient Heat Transfer Measurement in a Full-Stage Model Turbine,” *Journal of Turbomachinery*, vol. 111, no. 1, p. 20, 1989.
- [93] M. Collins, K. Chana, and T. Povey, “New technique for the fabrication of miniature thin film heat flux gauges,” *Measurement Science and Technology*, vol. 26, no. 2, p. 025303, 2015.
- [94] J. E. Doorly and M. L. G. Oldfield, “The theory of advanced multi-layer thin film heat transfer gauges,” *International Journal of Heat and Mass Transfer*, vol. 30, no. 6, pp. 1159–1168, 1987.
- [95] M. L. G. Oldfield, “Impulse Response Processing of Transient Heat Transfer Gauge Signals,” *ASME Journal of Turbomachinery*, vol. 130, no. April, p. 021023, 2008.

- [96] R. Abu Talib, *Detailed Investigation of the Low-Temperature Analogy of an Aircraft Engine Standard Fire-Test*. DPhil thesis, University of Oxford, 2003.
- [97] C. A. Neugebauer and M. B. Webb, “Electrical conduction mechanism in ultrathin, evaporated metal films,” *Journal of Applied Physics*, vol. 33, no. 1, pp. 412–499, 1962.
- [98] J. D. Wrbanek and K. L. H. Laster, “Preparation and Analysis of Platinum Thin Films for High Temperature Sensor Applications,” tech. rep., NASA Technical Memorandum, 2005.
- [99] M. Watanabe, D. R. Mumm, S. Chiras, and A. G. Evans, “Measurement of the residual stress in a Ptaluminide bond coat,” *Scripta Materialia*, vol. 46, no. 1, pp. 67–70, 2002.
- [100] U. Schmid and H. Seidel, “Influence of thermal annealing on the resistivity of titanium/platinum thin films,” *Journal of Vacuum Science & Technology A: Vacuum, Surfaces, and Films*, vol. 24, no. 6, pp. 2139–2146, 2006.
- [101] M. L. G. Oldfield, H. Burd, and N. Doe, “Design of wide-bandwidth analogue circuits for heat transfer instrumentation in transient tunnels,” *Heat and Mass Transfer in Rotating Machinery*, vol. -1, pp. 233–258, 1981.
- [102] R. T. Penty Geraets, M. McGilvray, L. Doherty, R. G. Morgan, C. M. James, T. Vanyai, and D. R. Buttsworth, “Development of a Fast-Response Calorimeter Gauge for Hypersonic Ground Testing,” *33rd AIAA Aerodynamic Measurement Technology and Ground Testing Conference*, pp. 1–18, 2017.
- [103] M. Schmitt, C. E. Frouzakis, Y. M. Wright, A. G. Tomboulides, and K. Boulouchos, “Investigation of wall heat transfer and thermal stratification under engine-relevant conditions using DNS,” *Special Issue Article International J of Engine Research*, vol. 17, no. 1, pp. 63–75, 2016.
- [104] A. Gafaar, *Combustor Turbulence Measurement Probe Development*. 4YP Report, University of Oxford, 2015.
- [105] D. R. Buttsworth, A. Agrira, R. Malpress, and T. Yusaf, “Simulation of Instantaneous Heat Transfer in Spark Ignition Internal Combustion Engines: Un-

- steady Thermal Boundary Layer Modelling,” in *ASME 2009 Internal Combustion Engine Division Fall Technical Conference*, pp. 413–419, ASME, 2009.
- [106] T. Clark, *The Development of a Diamond-Shielded Heat Transfer Gauge for Hypersonic Test Facilities*. Master’s thesis, University of Queensland, 2015.
- [107] E. Wörner, C. Wild, W. Müller-Sebert, R. Locher, and P. Koidl, “Thermal conductivity of CVD diamond films: high-precision, temperature-resolved measurements,” *Diamond and Related Materials*, vol. 5, no. 6-8, pp. 688–692, 1996.
- [108] B. Grandin and I. Denbratt, “The Effect of Knock on Heat Transfer in SI Engines,” in *SAE Technical Paper 2002-01-0238*, SAE International, 2002.
- [109] H. S. Carslaw and J. C. Jaeger, *Conduction of Heat in Solids*. Clarendon Press, 2nd ed., 1959.
- [110] S. J. Kline and F. A. McClintock, “Describing Uncertainties in Single Sample Experiments,” *Mechanical Engineering*, vol. 75, pp. 3–8, 1953.
- [111] I. Cresci, *High Pressure Nozzle Guide Vane Cooling System Flow Characteristics*. DPhil thesis, University of Oxford, 2017.
- [112] P. T. Ireland, *Internal Cooling of Turbine Blades*. DPhil thesis, University of Oxford, 1987.

Appendix A

Mass Flow Scaling Results for Convergent Geometry

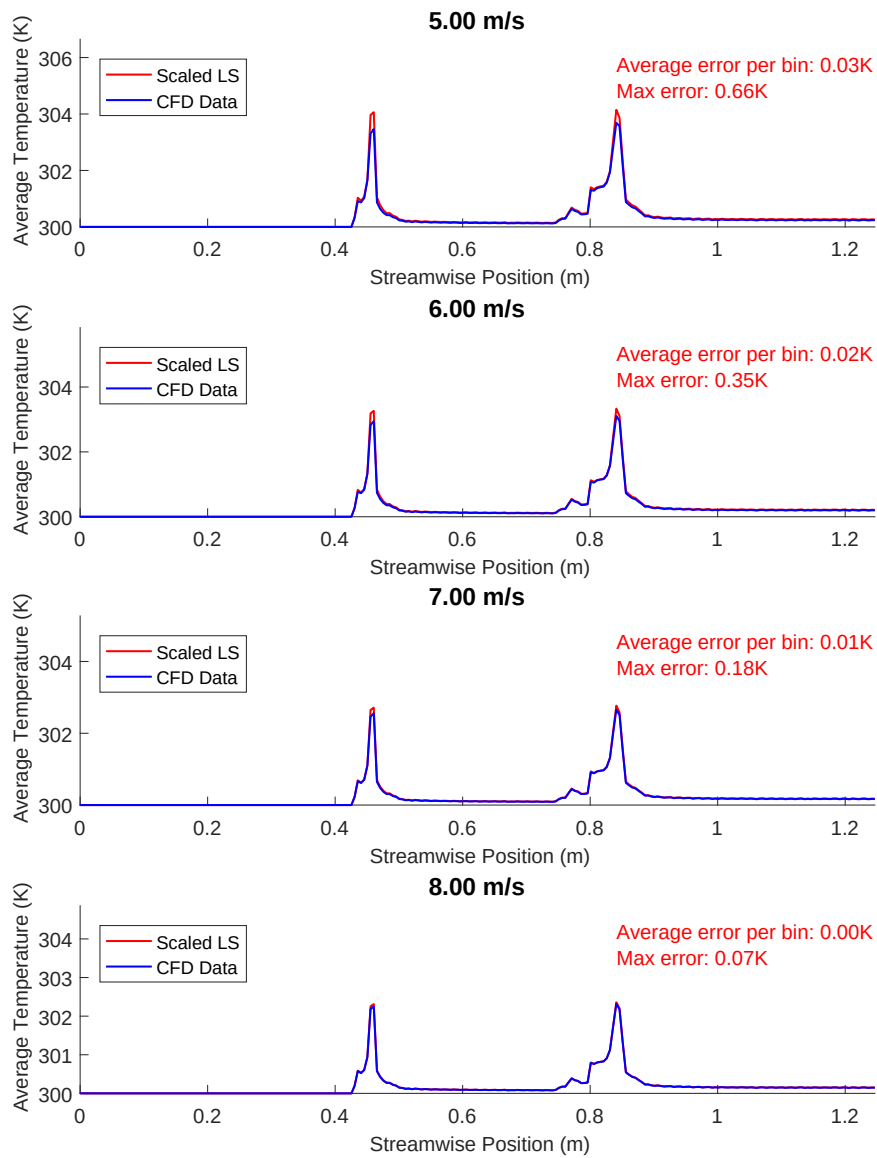


Figure A.1: Linear mass flow scaling applied to simplified convergent geometry for a 1 kW/m^2 heat flux boundary condition (scaled from a datum velocity of 9 m/s)

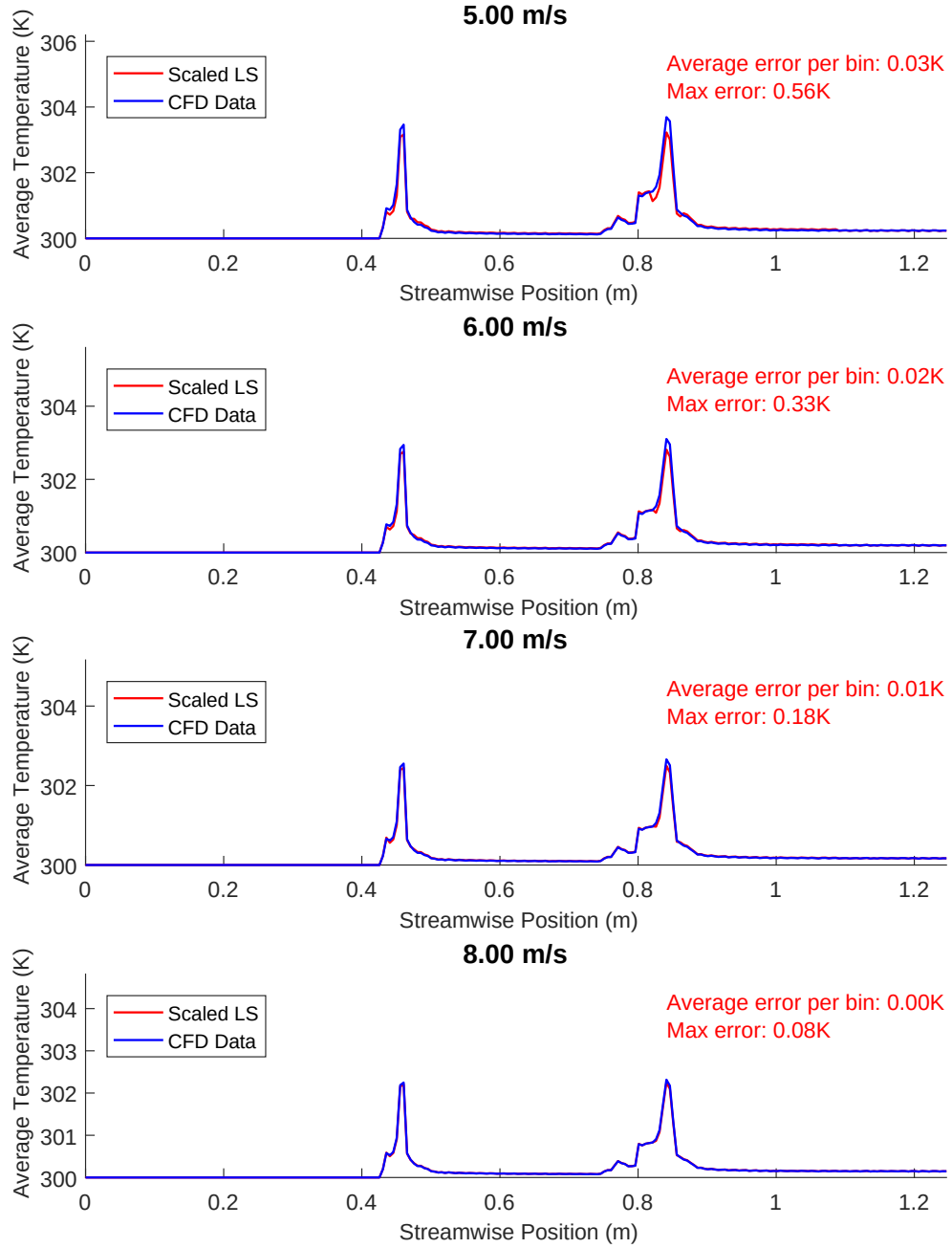


Figure A.2: Linear mass flow scaling with non-linear correction applied to simplified convergent geometry for a 1 kW/m^2 heat flux boundary condition (scaled from a datum velocity of 9 m/s)

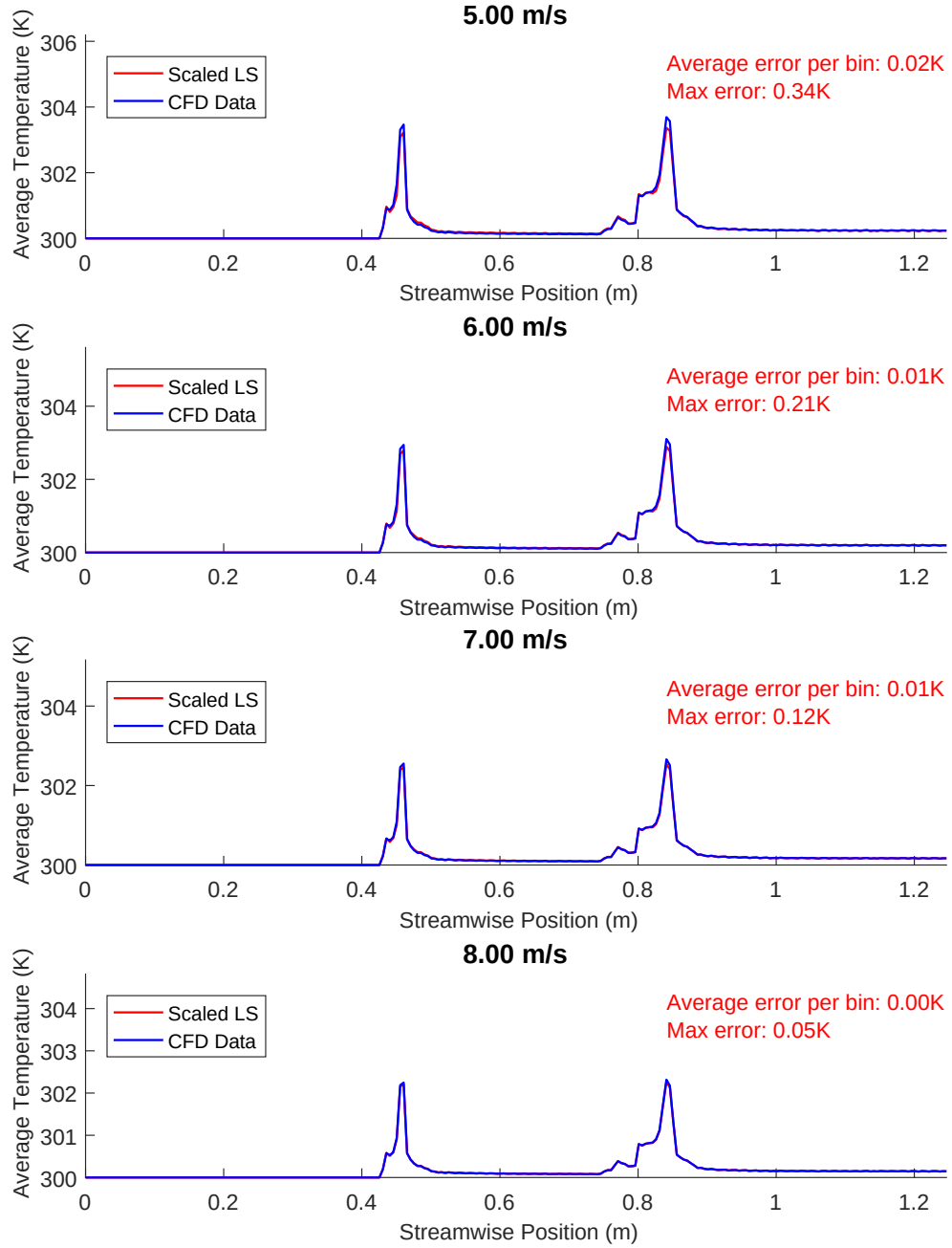


Figure A.3: Linear mass flow scaling with weighted non-linear correction applied to simplified convergent geometry for a 1 kW/m^2 heat flux boundary condition (scaled from a datum velocity of 9 m/s)

Appendix B

Thin Film Gauge Calibration Results

B.1 Calibration of vitreous enamel coated test disc

Table B.1: Calibration of thin film gauges on vitreous enamel test disc

Gauge Number	α value	R_0 value (Ω)	α 95% confidence value	R_0 95% confidence value (Ω)
1	0.0019	31.37	0.000063	0.090
2	0.0020	18.58	0.000005	0.004
3	0.0019	26.60	0.000045	0.055
4	0.0022	42.12	0.000005	0.009

B.2 Calibration of alumina-zirconia coated test disc

Table B.2: Calibration of thin film gauges on raw alumina-zirconia test disc

Gauge Number	α value	R_0 value (Ω)	α 95% confidence value	R_0 95% confidence value (Ω)
1	0.0026	51.32	0.000009	0.021
2	0.0019	53.08	0.000299	0.746
3	0.0020	57.52	0.000323	0.888
4	0.0024	39.66	0.000019	0.036
5	0.0025	44.00	0.000022	0.046

Table B.3: Calibration of thin film gauges on smoothed alumina-zirconia test disc

Gauge Number	α value	R_0 value (Ω)	α 95% confidence value	R_0 95% confidence value (Ω)
1	0.0026	50.28	0.000030	0.073
2	0.0026	49.64	0.000023	0.054
3	0.0026	49.96	0.000031	0.074
4	0.0026	51.11	0.000031	0.076
5	0.0026	45.89	0.000032	0.071

B.3 Calibration of ceramic substrate test samples

Table B.4: Calibration of thin film gauges on ceramic samples

Ceramic Substrate	α value	R_0 value (Ω)	α 95% confidence value	R_0 95% confidence value (Ω)
Fused Quartz	0.0024	94.51	0.000005	0.002
Alumina	0.0024	54.62	0.000003	0.007

B.4 Calibration of engine probes

Table B.5: Calibration of sample probes for engine testing

Probe Number	Probe Type	α value	R_0 value (Ω)	α 95% confidence value	R_0 95% confidence value (Ω)
22	Diamond	0.0024	95.98	0.000286	0.013
25	Ceramic	0.0026	43.76	0.000003	0.007
26	Ceramic	0.0025	73.03	0.000015	0.005

Appendix C

Uncertainty Analysis

Estimates of the uncertainty for the experimental measurements in this work has been investigated using the perturbation method outlined by Kline and McClintock [110].

If the result R of an experiment is calculated from a set of measurements X_1 to X_N , it can be represented using Equation C.1.

$$R = R(X_1, X_2, X_3, \dots, X_N) \quad (\text{C.1})$$

The effect of the uncertainty in a single measurement X_i on a calculated result R can be represented using Equation C.2.

$$\delta R_{X_i} = \frac{\partial R}{\partial X_i} \delta X_i \quad (\text{C.2})$$

The rate of change in the measured value $\frac{\partial R}{\partial X_i}$ can be approximated to a linear change caused by a small perturbation in the measurement ΔX_i and is calculated using the central difference method.

$$\frac{\partial R}{\partial X_i} = \frac{R(X_i + \Delta X_i) - R(X_i - \Delta X_i)}{2\Delta X_i} \quad (\text{C.3})$$

Kline and McClintock [110] showed that the overall uncertainty in the result R is best calculated using the root-sum-square of the component uncertainties. The relative uncertainty is calculated by dividing each individual contribution by R^2 . Hence, for a result R , which is a function of several independent measurements, the relative uncertainty ϵ can be calculated using Equation C.4

$$\epsilon = \left[\sum_{i=1}^N \left(\frac{\partial R}{\partial X_i} \frac{\Delta X_i}{R} \right)^2 \right]^{0.5} \quad (\text{C.4})$$

All uncertainties listed below are specified by the manufacturer unless otherwise stated.

C.1 Dimensionless Temperature Coefficient

The dimensionless temperature coefficient is calculated using the following formula:

$$\Phi = \frac{T_i - T_{in}}{\frac{Q}{\dot{m}c_p}} \quad (\text{C.5})$$

The uncertainty in the measurement of the mass flow rate was reported to be 6% for a similar setup by Cresci [111]. The uncertainty in the $T_i - T_{in}$ term was assessed using the 95% confidence interval method (assuming the samples were drawn from a normal distribution) as this quantity was calculated as the average of multiple samples.

The uncertainty in the measurement of the dimensionless temperature parameter can be calculated from the root-sum-square of the following three measured quantities.

$$\epsilon_{\Phi} = \sqrt{\epsilon_Q^2 + \epsilon_{\dot{m}}^2 + \epsilon_{(T_i - T_{in})}^2} \quad (\text{C.6})$$

Table C.1: Uncertainty for dimensionless temperature coefficient parameters

Parameter	Value	Effect on Φ
Q	$\pm 0.5W$	1.1%
$\dot{m}$	$\pm 6\%$	5.7%
$T_i - T_{in}$	$\pm 0.002\%$	0.1%

Using the parameters from Table C.1, the relative uncertainty in the dimensionless temperature parameter was shown to be approximately 5.9%.

C.2 Heat Transfer Coefficient Measurements

The local heat transfer coefficients on the exhaust part are calculated using the following formula:

$$T_s = T_0 + (T_g - T_0)[1 - \exp(\beta^2) \operatorname{erfc}(\beta)] \quad (\text{C.7})$$

where

$$\beta = \frac{h\sqrt{t}}{\sqrt{\rho c_p k}} \quad (\text{C.8})$$

Ireland reported the thermal product ($\sqrt{\rho c k}$) of Perspex to be $569 \pm 29 Jm^2\sqrt{K}$ [112]. The uncertainty in the measurement of the time parameter was estimated to be 0.4 seconds. Hence, the relative uncertainty of the local heat transfer coefficient measurement can be calculated from the individual contributions of five measured quantities.

$$\epsilon_h = \sqrt{\epsilon_{\sqrt{\rho c k}}^2 + \epsilon_{T_s}^2 + \epsilon_{T_0}^2 + \epsilon_{T_g}^2 + \epsilon_t^2} \quad (\text{C.9})$$

Table C.2: Uncertainty for heat transfer coefficient parameters

Parameter	Value	Effect on h
$\sqrt{\rho c k}$	$\pm 29 Jm^2\sqrt{K}$	5.1%
T_s	$\pm 0.25^\circ\text{C}$	2.4%
T_0	$\pm 0.5^\circ\text{C}$	3.5%
T_g	$\pm 0.5^\circ\text{C}$	1.2%
t	$\pm 0.4\text{sec}$	0.7%

Using the values from Table C.2, the relative uncertainty in the measurement of the local heat transfer coefficients on the exhaust part was estimated to be 6.8%.

C.3 Surface Temperature Measurements using Thin Film Gauges

The surface temperature measurement is calculated from the resistance of the thin film gauge. As the thin film gauge is supplied with a constant current, the resistance is calculated from the ratio of voltage to current.

$$T_s = T_0 + \frac{1}{\alpha_{tc}} \left(\frac{R}{R_0} - 1 \right) \quad (\text{C.10})$$

where:

$$R = \frac{V}{I} \quad (\text{C.11})$$

The uncertainties in the α_{tc} and R_0 parameter are calculated during the calibration of the thin film gauges using the method outlined in Section 6.4. The uncertainty in the surface temperature measurement is calculated from the component uncertainties of the following measured quantities.

$$\epsilon_{T_s} = \sqrt{\epsilon_{\alpha_{tc}}^2 + \epsilon_{R_0}^2 + \epsilon_V^2 + \epsilon_I^2} \quad (\text{C.12})$$

Table C.3: Uncertainty for surface temperature parameters

Parameter	Value	Effect on T_s
α_{tc}	$\pm 0.2\%$	0.2%
R_0	$\pm 0.05\%$	0.2%
V	$\pm 0.5\%$	1.7%
I	$\pm 0.5\%$	1.7%

Using the values in Table C.3, the uncertainty in the surface temperature measurements was calculated to be approximately 2.4%.