On CFD-Analysis of Heat Transfer of a Heavy-Duty Diesel Engine

Helgi Skuli Fridriksson

Thesis for the degree of Licentiate of Engineering, 2011
To my family
Abstract

In this work, a heavy-duty diesel engine was studied by employing CFD simulations on a closed volume engine segment. These simulations were used to evaluate both the effects from certain parameters on the wall heat transfer, and to examine how reduction of heat transfer would effect the engine performance and emission levels.

In the first study, an initial investigation concerning the in-cylinder heat transfer was performed. This included a simple parameter study, performed to estimate the contribution of each parameter on the in-cylinder engine heat transfer. The parameter study revealed that most of the parameters selected do in-fact significantly affect the in-cylinder heat transfer. However, the effects on the indicated mean effective pressure, or indicated power output are different. Changing the amount of EGR and the swirl number did not show any dramatic effects on the indicated power output, while the injection duration showed much more dynamic effects on the indicated power output.

In the second study, an investigation was made on the same heavy duty diesel engine, using both conventional diesel combustion mode and a partially premixed combustion (PPC) mode. Both combustion modes were validated using experimental data, before various heat flux boundary conditions were applied. These conditions were used to evaluate the engine response in terms of engine performance and emission levels for the different levels of heat rejection. The engine performance was measured in terms of specific fuel consumption and estimated power output, while the calculated net soot and accumulated NO\textsubscript{x} mass fractions were used for comparing the emission levels. The results showed improved efficiency for both combustion types, but only the PPC combustion mode managed that without increasing the production of NO\textsubscript{x} emissions severely. It was also noticed that the emission levels, for the baseline cases, were much lower for the PPC combustion mode than for the diesel combustion mode.
List of publications

Publications within this thesis work:


Publications outside the scope of this thesis work:

Nomenclature

Latin letters

B  Engine bore [m]
C  Model constant (various models)
cp  Specific heat
D  Characteristic length (equal to engine bore)
f  Elliptic relaxation function
H  Total enthalpy
h  Heat transfer coefficient [W/m²K]
k  Turbulent kinetic energy
kij  Reaction rate constant
P  Pressure [bar]
q  Heat flux [W/m²]
Sp  Mean piston velocity
jfua  Reaction rate of fuel in the eddy break-up model
T  Temperature [K]
U  Velocity
V  Volume
Vd  Displacement volume
yi  Concentration of species i

Greek letters

αs  Scaling factor in global heat transfer models
β  Velocity scaling in the Hohenberg model
Γ  Compound wall function exponent
δij  Kronecker delta
ε  Turbulence dissipation
ζ  Velocity scale ratio
κ  Von Karman constant [-]
µ  Dynamic viscosity
ν  Kinematic viscosity
ρ  Density
τij  Viscous stress
τR  Chemical reaction time scale
Φ  Equivalence ratio [-]
ω  Angular velocity

Acronyms & Abbreviations

ATDC  After Top Dead Center
BSFC  Break specific fuel consumption
BTDC  Before Top Dead Center
CFD  Computational Fluid Dynamics
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ECFM-3Z</td>
<td>3 Zone Extended Coherent Flame Model</td>
</tr>
<tr>
<td>EGR</td>
<td>Exhaust Gas Recirculation</td>
</tr>
<tr>
<td>EVM</td>
<td>Eddy Viscosity Model</td>
</tr>
<tr>
<td>IC</td>
<td>Internal Combustion</td>
</tr>
<tr>
<td>LHR</td>
<td>Low Heat Rejection</td>
</tr>
<tr>
<td>LTC</td>
<td>Low Temperature Combustion</td>
</tr>
<tr>
<td>MEP</td>
<td>Mean Effective Pressure</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>PPC</td>
<td>Partially Premixed Combustion</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Average Navier-Stokes</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>RoHR</td>
<td>Rate of heat release</td>
</tr>
</tbody>
</table>
Acknowledgements

In the following paragraphs, people who have helped me along the way to my degree of Licentiate of Engineering are acknowledged. To those of you that feel you are missing from the list, I am no less grateful.

I would like to thank my main supervisor, Professor Bengt Sundén, for the opportunity of pursuing my PhD studies at the Department of Energy Sciences and the freedom within my work. I think of myself as very fortunate to be able to pursue a PhD degree in such a fascinating field as heat transfer.

To my assistant supervisors, Dr. Shahrokh Hajireza and Assistant Professor Martin Tunér, I am also very grateful. We have had quite a few inspiring discussions and I have received valuable help that has aided my development as a researcher. Critical thinking and improved writing skills is something I have developed under your guidance.

My research project is one out of three projects within a project cluster called D60, comprising PhD students and supervisors from three different divisions. I would like to thank my fellow researchers for the interesting discussions and hope that the co-operation between the different divisions will continue to blossom as D60 continues.

This thesis work would not have been possible without the financial support of the Swedish Energy Agency, which finances the D60 project.

In my work I have been in contact with employees from different car manufactures in Sweden, especially Scania and Volvo, who have helped with engine geometries and boundary conditions for my simulations. This has helped my tremendously in my analysis of the engine heat transfer.

It would be difficult to perform engine simulations without a CFD tool. This has graciously been provided by AVL List GMBH, in Austria.

I would like to thank the entire staff at the Department of Energy Sciences for a very good work environment. To my friends and fellow PhD students at the Division of Heat Transfer, Andreas, Hedvig, Henrik, Martin, Munir, Tareq, Wamei and Zahra, I am extremely grateful for all your cooperation and support throughout this period.

To my parents, brother and sister, I am very grateful for all the support I have received, regardless of what I have decided to do. You have taught me never to give up and always keep on trying.

Finally, I want to thank my wife, Berglind, and our children, Bírta and Daniel, for pushing me forwards and always being supportive, both in my work and other activities in my life.
## Contents

1 Introduction ................................................. 1  
  1.1 Motivation .............................................. 1  
  1.2 Importance of engine heat transfer ......................... 2  
  1.3 Low heat rejection engines ................................. 3  
  1.4 Low temperature combustion ................................ 4  
  1.5 Temperature dependent emissions ........................... 5  
  1.6 The D60 project ......................................... 6  

2 Theoretical Background ................................. 7  
  2.1 Engine heat transfer simulations ............................ 7  
  2.2 Modeling procedures ..................................... 8  
    2.2.1 Global models ....................................... 8  
    2.2.2 Multidimensional models ............................. 9  
  2.3 Current work ........................................... 10

3 Model Set-up ............................................ 12  
  3.1 Turbulence model ....................................... 12  
    3.1.1 Direct Numerical Simulation .......................... 12  
    3.1.2 Large Eddy Simulation ............................... 13  
    3.1.3 RANS Simulation .................................... 13  
  3.2 Combustion model ....................................... 14  
    3.2.1 Eddy break-up model ................................ 14  
    3.2.2 ECFM-3Z model ..................................... 15  
  3.3 Emission models ........................................ 15  
    3.3.1 NO\textsubscript{x} modeling .......................... 15  
    3.3.2 Soot modeling ...................................... 16  
  3.4 Spray behavior ......................................... 16  
  3.5 Wall heat transfer ..................................... 17  
    3.5.1 "Standard" temperature wall function ................. 17  
    3.5.2 Han & Reitz temperature wall function ............... 17

4 Studied Cases ........................................... 19  
  4.1 Engine geometry ....................................... 19  
    4.1.1 Boundary & initial conditions ....................... 20  
  4.2 Parameter study ........................................ 20  
  4.3 Heat rejection study ................................... 21

5 Results & Discussions ................................. 23  
  5.1 Baseline validation .................................... 23
5.1.1 Diesel combustion ........................................... 23
5.1.2 Partially premixed combustion ............................ 24
5.2 Parameter study ................................................ 26
  5.2.1 EGR variation .............................................. 26
  5.2.2 Injection duration variation .............................. 28
  5.2.3 Swirl number variation ................................... 30
5.3 Heat rejection study .......................................... 32
  5.3.1 Diesel combustion ....................................... 32
  5.3.2 Partially premixed combustion ......................... 36

6 Conclusions and Future Work ............................... 39

A Correlations for Global Models ............................. 47
  A.1 Annand & Ma ............................................... 47
  A.2 Woschni .................................................... 47
  A.3 Hohenberg ................................................. 48
  A.4 Chang ....................................................... 48

B Important Governing Equations ............................. 50
  B.1 Flow equations ............................................. 50
  B.2 Turbulence model equations ............................... 51
  B.3 Wall treatment ............................................. 51
    B.3.1 Standard velocity wall functions ...................... 51
    B.3.2 Compound wall functions ............................. 52
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Energy cascade from fuel energy to the engine’s power output</td>
<td>2</td>
</tr>
<tr>
<td>1.2</td>
<td>$\Phi - T$ map constructed by Kamimoto and Bae [24]</td>
<td>5</td>
</tr>
<tr>
<td>3.1</td>
<td>The energy cascade of turbulent flow [54]</td>
<td>13</td>
</tr>
<tr>
<td>4.1</td>
<td>Computational domain shown at 20° BTDC. The grid has around 113,000 and 66,000 cells at BDC and TDC, respectively</td>
<td>20</td>
</tr>
<tr>
<td>5.1</td>
<td>Mean Pressure trace from experiments and CFD simulations</td>
<td>24</td>
</tr>
<tr>
<td>5.2</td>
<td>Accumulated heat release and heat release rate from experiments and CFD simulations</td>
<td>24</td>
</tr>
<tr>
<td>5.3</td>
<td>Mean Pressure trace from experiments and CFD simulations</td>
<td>24</td>
</tr>
<tr>
<td>5.4</td>
<td>Accumulated heat release and heat release rate from experiments and CFD simulations</td>
<td>25</td>
</tr>
<tr>
<td>5.5</td>
<td>Mean pressure of the EGR variation</td>
<td>26</td>
</tr>
<tr>
<td>5.6</td>
<td>Mean temperature and heat release rates for the EGR parameter sweep</td>
<td>27</td>
</tr>
<tr>
<td>5.7</td>
<td>Wall heat fluxes for the EGR variation</td>
<td>27</td>
</tr>
<tr>
<td>5.8</td>
<td>Mean pressure of the injection duration variation</td>
<td>28</td>
</tr>
<tr>
<td>5.9</td>
<td>Mean temperature and heat release rates for the injection duration parameter sweep</td>
<td>28</td>
</tr>
<tr>
<td>5.10</td>
<td>Wall heat fluxes for the injection duration variation</td>
<td>29</td>
</tr>
<tr>
<td>5.11</td>
<td>Mean pressure of the swirl number variation</td>
<td>30</td>
</tr>
<tr>
<td>5.12</td>
<td>Mean temperature and heat release rates for the swirl number parameter sweep</td>
<td>30</td>
</tr>
<tr>
<td>5.13</td>
<td>Wall heat fluxes for the swirl number variation</td>
<td>31</td>
</tr>
<tr>
<td>5.14</td>
<td>Heat release from combustion for different heat rejection levels</td>
<td>32</td>
</tr>
<tr>
<td>5.15</td>
<td>Engine performance for different heat rejection levels</td>
<td>33</td>
</tr>
<tr>
<td>5.16</td>
<td>Emission levels for different heat rejection levels</td>
<td>33</td>
</tr>
<tr>
<td>5.17</td>
<td>Average temperature in the cylinder for different heat rejection levels</td>
<td>34</td>
</tr>
<tr>
<td>5.18</td>
<td>Equivalence ratio for the diesel combustion, from TDC to 50 CAD ATDC</td>
<td>34</td>
</tr>
<tr>
<td>5.19</td>
<td>Cylinder gas temperature for the diesel combustion, from TDC to 50 CAD aTDC</td>
<td>35</td>
</tr>
<tr>
<td>5.20</td>
<td>Heat release from combustion for different heat release levels</td>
<td>36</td>
</tr>
<tr>
<td>5.21</td>
<td>Engine performance for different levels of heat rejection</td>
<td>36</td>
</tr>
<tr>
<td>5.22</td>
<td>Emission mass fractions for different levels of heat rejection</td>
<td>37</td>
</tr>
<tr>
<td>5.23</td>
<td>Average temperature in the cylinder for different heat rejection levels</td>
<td>37</td>
</tr>
<tr>
<td>5.24</td>
<td>Equivalence ratio for the PPC combustion, from TDC to 50 CAD ATDC</td>
<td>38</td>
</tr>
</tbody>
</table>
5.25 Cylinder gas temperature for the PPC combustion, from TDC to 50 CAD aTDC. 38

B.1 Mean velocity ($u^+$) distribution in the turbulent boundary layer. 52
Chapter 1

Introduction

This thesis work has been carried out at the Division of Heat Transfer within the Faculty of Engineering of Lund University. It is a part of the Swedish Energy Agency’s project D60, which is described at the end of the chapter.

1.1 Motivation

The lifestyle within the modern western society is highly dependent on transportation, weather it is by personal cars, trains, airplanes or other transportation means. More or less all of the modern transportation means are currently considerable sources of greenhouse gas emissions. Numbers from the Swedish Energy Agency state that the transport sector in Sweden was responsible for roughly 26% of the national energy use in the year 2008. Out of these 26%, petrol and diesel encompassed roughly 89% of the energy requirement for the domestic transport [1].

The situation is similar in most of the developed countries, so the dependence of the transport sector on fossil fuels is clear.

The amount of CO$_2$ emissions per energy unit is relatively high from fossil fuels, which is not desirable in a global climate perspective, and it has to be reduced. One efficient way of reducing these emissions would be to replace the fossil fuels with other fuels, such as biofuels. The major drawback of the biofuels, however, is their production costs, which is considerably higher than those for fossil fuels [2]. Additionally, the available biofuel techniques and raw material today cannot be used to produce sufficient amount of fuel to entirely phase out fossil fuels [3].

An alternative possibility is to find ways to increase the efficiency of the current internal combustion (IC) engines, leading to less CO$_2$ emission for each unit volume of fuel.
1. Introduction

1.2 Importance of engine heat transfer

When discussing diesel engine heat transfer, it should be noted that a diesel engine can be divided into a series of interacting subsystems, as pointed out by Borman & Nishiwaki [4]. They list six subsystems within the engine, each with their own level of complexity when it comes to the evaluation of flow patterns and heat transfer. In each of the subsystems heat transfer plays a significant role for the performance of the engine. During the intake stroke higher temperatures will decrease the volumetric efficiency, indicating that less amount of fresh air will be used in the cycle, and thereby slightly lowering the overall performance of the engine. During compression the peak pressure is affected by the heat transfer mechanisms in the engine cylinder. However, the largest and most significant contribution from the heat transfer process in a diesel engine is experienced during the combustion and expansion. Then the convective and radiative heat transfer from the gases and particles in the cylinder is combined with the conductive heat transfer through the cylinder head, walls and piston to the cooling passages and lubricating oil [4, 5]. Due to the transient behavior of the fluid motion inside the engine cylinder, the heat flux to the solid components of the cylinder may vary from 0 to as much as 10 MW/m$^2$ during the span of a few milliseconds. Furthermore, two points on the cylinder wall, separated only by one centimeter, may experience the same difference in heat flux, introducing large thermal stresses in the solid components [4, 6]. Heat transfer is also important for spray and combustion related phenomena, such as droplet evaporation, autoignition and flame-wall interaction [7].

![Figure 1.1: Energy cascade from fuel energy to the engine’s power output.](image)

Figure 1.1 shows the energy cascade from fuel energy down to the power output for a heavy duty diesel engine. In the case presented here the FuelMEP was estimated to a value around 63 bar, IMEPg to 29 bar and BMEP to 27 bar, yielding just under 43% break efficiency. The figure shows all the major losses within the engine cylinder, given in MEP (Mean Effective Pressure). There are five main sources of energy losses in the engine cylinder. These are:
1.3 Low heat rejection engines

- Losses due to incomplete combustion, \( Emission\ MEP \approx 1 \) bar.
- Losses due to heat transfer \( \approx 33 \) bar.
  - Heat losses through solid engine components, \( Heat\ Transfer\ MEP \).
  - Heat associated with the exhaust gases, \( Exhaust\ MEP \).
- Losses experienced while pumping out the exhaust gases and replacing them with a fresh gas charge, \( Pump\ MEP \approx 1.5 \) bar.
- Frictional losses in the system, \( Friction\ MEP \approx 0.5 \) bar.

From the figure it is clear that heat losses are the dominant source of losses in an engine. According to the second law of thermodynamics, the entire energy content of the fuel cannot be converted into useful energy, which implies that some energy must be lost in form of heat. It is, however, preferable that the amount of energy lost to heat is kept at a minimum level in order to maintain high overall efficiency of the engine. For this reason, it is of great importance to have control of the heat losses in the system when designing an internal combustion engine. This implies that the knowledge on temperature distribution and heat losses inside and around the engine cylinder must be increased. This increased knowledge can also be used to ensure that the remaining heat is directed through the exhaust system. By doing that, the waste heat can be utilized in a waste heat recovery system. Waste heat recovery is not a subject in this thesis work, but it is important to point out that it is widely researched.

In order to reduce heat losses in the engine there are two main alternatives. The first alternative, "low heat rejection", has been researched for a number of years. It is comprised of various insulation techniques for insulation of various components of the engine cylinder. The second alternative, "low temperature combustion", is relatively recent and mainly aims to lower the peak temperatures in the engine cycle. Both of these alternatives are described in more details in the following sections.

1.3 Low heat rejection engines

Engines that have been partly coated by a thermally insulated material are called low heat rejection engines. An engine that has been thermally insulated, meaning that no heat is transferred through the engine walls, is called an adiabatic engine. The term adiabatic engine is, however, merely a theoretical term because no real engine has shown a fully adiabatic behavior.

Historically, some research has been done on low heat rejection (LHR) or even adiabatic engines, both numerically and experimentally, as shown in [8, 9]. The results have been quite diverse. The numerical results usually indicate a positive response in engine performance with increased insulation. Experimental results have, however, not been as favorable, because a few researchers have shown little or no efficiency gains with LHR engines. What this difference between experiments and simulations is based on is not clear, but it might be based on how near
wall conditions are handled in the simulation work and how the experiments are performed and analyzed. Performing simulations and experiments on the same engine at the same load point would be beneficial to examine these differences, but this has rarely been done in the examined literature. It has been shown that by changing parameters in the engine configuration, e.g., injection characteristics [10, 11] or compression ratio [12], favorable results can be achieved. This is mainly due to the effects these changes have on the combustion process, often leading to a leaner, lower temperature combustion, as described in the next section. Other research works [13–15] have shown potential efficiency gains using a LHR engine, both theoretically and experimentally. Tunér [16] has demonstrated that with the introduction of the new low temperature combustion concepts, the LHR engine is an attractive option.

1.4 Low temperature combustion

As stated before, it is of interest to reduce the heat losses in the engine in the attempt to achieve higher mechanical work output. The reduction of heat losses through walls will result in elevated combustion temperatures and elevated exhaust temperatures [15]. The raised temperatures might introduce increased local emission production in the cylinder, which is not desired due to strict emission regulations as well as efficiency penalties from exhaust gas after-treatment. One way to circumvent this problem is to incorporate some of the relatively newly introduced Low Temperature Combustion, or (LTC), strategies. LTC engines are a group of engines that are based on lean- or diluted mixtures of fuel and air resulting in lower peak temperatures during combustion and was first introduced with the 2-stroke ATAC engine, presented by Onishi et al. [17]. This lean-burn autoignition concept was revived in the late 90’s and became better known as HCCI, or Homogeneous Charge Compression Ignition, where a homogeneous charge of fuel and air is ignited by compression, as described in [18, 19]. The problem with HCCI combustion has been related to combustion control, due to the nature of the combustion concept, which has been overcome by the introduction of a concept called Partially premixed combustion, or PPC. Compared to HCCI, the load range for PPC is considerably wider, without compromising the performance of the engine. The two concept share the property of higher thermal efficiency, compared to traditional SI and CI engines, together with low emission levels [18–20].

The PPC concept, as described by Manente [20], is an intermediate process between HCCI combustion, using a fully premixed charge, and classical diesel combustion, where fuel is injected while combustion is occurring. Fuel is thereby injected in the compression stroke to allow for some mixing with air before the start of combustion. According to Manente [20], all of the fuel should be injected before the start of combustion, but not too early in order to prevent violent combustion with high peak pressure, peak pressure rates and pressure oscillations. This fuel injection strategy is combined with high levels of EGR in order to dilute the air/fuel mixture in order to assure low temperature combustion. This strategy has shown to provide high efficiency and extremely low emissions for a wide load range of a heavy duty diesel engine [21].
A similar combustion strategy has been developed by Kokjohn et al. at the University of Wisconsin, which they have chosen to call PCCI, or Premixed Charge Compression Ignition. This concept, along with a duel fuel system, has also shown impressive efficiency gains over a wide load range, compared to traditional CI engines [22].

1.5 Temperature dependent emissions

The global chemical reaction for a complete combustion of a hydrocarbon based fuel is given by Eq. (1.1), which shows that the amount of CO$_2$ in the exhaust is directly proportional to the amount of fuel used during combustion [5].

$$C_a H_b + \left( a + \frac{b}{4} \right) \left( O_2 + 3.773 N_2 \right) \rightarrow aCO_2 + \frac{b}{2} H_2O + 3.773 \left( a + \frac{b}{4} \right) N_2 \tag{1.1}$$

Due to the physical behavior of the diesel spray, there is always a local rich zone and a local lean zone in a diesel combustion. In the rich zone emissions, like unburnt hydrocarbon (UHC) and carbon monoxide (CO), are formed while in the lean zone excess oxygen is available. While the previously mentioned emissions are mostly controlled by the air/fuel ratio, other emissions such as NO$_x$ and soot are also temperature dependent. NO$_x$ is formed in lean mixtures at higher temperatures, while soot is formed in rich mixtures. Akihama et al. [23] showed, by expanding the $\Phi - T$ map introduced by Kamimoto and Bae [24] in 1988, that a rich diesel combustion can be used without producing any soot and NOx as long as the temperature is kept low. Looking at the $\Phi - T$ map, shown in Fig. 1.2, it can be said that low temperature combustion is essential to the reduction of NOx and soot emissions.

![Figure 1.2: $\Phi - T$ map constructed by Kamimoto and Bae [24].](image-url)
1. Introduction

1.6 The D60 project

This thesis is a part of the outcome of a research project which is carried out at the department of Energy Sciences at Lund University. Within the project the heat transfer mechanisms inside and around an engine cylinder are studied. The project, named *D60: The 60% efficient Diesel Engine*, is funded by the Swedish Energy Agency and aims to improve the efficiency of a diesel engine drastically, hence the name. Increasing the efficiency will directly reduce the emissions of one of the major green house gases, CO$_2$, because the engine will consume less fuel for the same work load. In order to achieve these ambitious goals, a better understanding of the heat transfer mechanisms inside and around the engine cylinder is critical. When the knowledge on the in-cylinder processes has been improved, possible solutions to improve efficiency can be made. It is unlikely that conventional diesel combustion mode will be able to deliver 60% efficiency. Therefore, it is of interest to examine how low temperature combustion concepts can aid in the development towards higher efficiency.

The project is divided into three parts. The first part is carried out at the *division of Combustion Engines* and includes experimental work on a single-cylinder diesel engine along with one dimensional system simulation. The second part, carried out at the *division of Combustion Physics*, is closely tied to the first one because it involves laser based diagnostics for temperature estimation inside the engine cylinder. The third part, carried out at the *division of Heat Transfer*, includes three dimensional engine simulations for heat transfer analysis. This part will, when calibrated with the experiments, give more detailed spatial and temporal resolution on important parameters, such as temperature and local emission production.
Chapter 2

Theoretical Background

As the work for this thesis is focused on heat transfer analysis, the background coverage will also be focused on heat transfer applications rather than engine simulations in general. In this chapter three questions will be answered, why, how and what. In the first section the question of why engine heat transfer simulations are performed will be answered. The second section will show how can it be done, and finally the third section will state what has been done in this thesis work.

2.1 Engine heat transfer simulations

The automotive industry has traditionally, like many other large industries, relied heavily on experimental work and prototype testing in order to achieve a better product. This experimental mentality is still quite dominating, but now in a combination with various simulation technologies. The advantage of experimental work is that the person performing the experiment is physically in control of the engine. However, the measurements are only as reliable and accurate as the measurement equipment and how well one can control and adjust the equipment. Additionally, the results presented from experimental work is extremely sensitive to the quality of the analysis performed on the raw experimental data. The largest drawback of an engine experiment is the lack of access, i.e., not being able to measure everything without compromising material strength or interfering with the natural flow field at the measurement location. With engine simulations one can, based on the model used, obtain information on almost any parameter at any operating condition, including extreme conditions not available in an experimental setup. The major problem with simulation work is that it is bound to simplifications of the physical behavior and/or geometry, because the resolving the smallest length scales in an engine cylinder would require an enormously fine computational grid. These simplifications call for a series of sub-models that represent the physical behavior of the simplified process, for instance a fuel spray. In order to obtain a realistic solution to the problem, these sub-models must be adjusted to accurately describe reality.
This is done by calibrating the simulation model to one or several experimental cases with the same boundary and operating conditions.

2.2 Modeling procedures

How can engine heat transfer be modeled? There are numerous alternatives, but according to Borman & Nishiwaki, who published a review on internal combustion engine heat transfer in 1987 [4], the available heat transfer models can be divided into five main groups, namely

1. Global (one-zone) thermodynamics models
2. Zonal (more than one zone) thermodynamics models
3. One-dimensional analytical and numerical fluid-dynamics models
4. Multidimensional numerical fluid dynamics models (CFD models)
5. Radiation heat-transfer models

The group containing one-dimensional models, where the energy equation is solved only in one dimension, has not been used lately to a large extent in the automotive industry. The last group, radiation models, are now usually included in CFD models. This reduces the classification of widely used heat transfer models from five to three main categories, global models, zonal models and CFD models.

The most common use of zonal models in the automotive industry today is for combustion modeling, whereas heat transfer is mostly modeled by either global models or multidimensional CFD models. The zonal models have therefore not been investigated for the purpose of this thesis work.

2.2.1 Global models

Since the early days of combustion engine development, experimental data has been used for heat transfer analysis. A large group of models, named global models, is based on empirical or semi-empirical formulations for the mean, crank angle dependent, heat transfer coefficients. Within this group only one instantaneous heat transfer coefficient is calculated for the cylinder as a whole, i.e., piston, head and cylinder liner all share the same coefficient at a given time. The first model of this kind was proposed by Nusselt in 1923 [25], which was based on experiments in a spherical bomb. His correlation, which contained both convective and radiant heat transfer, was a function of the mean piston speed, pressure trace, gas and wall temperatures as well as the emissivities of gas and wall. Although Nusselt’s heat transfer coefficient was supposed to predict the time average heat flux, it has more often been used for the prediction of the instantaneous heat flux.

Another empirical correlation of the total, instantaneous heat transfer coefficient was proposed in 1939 by Eichelberg [26]. His correlation was based on experiments on naturally-aspirated diesel engines and given as a function of the mean piston speed, pressure and temperature.
Following the work of Nusselt and Eichelberg, some authors turned to the use of similarity laws of steady turbulent heat transfer. The most widely known correlations of that form are those of Annand & Ma [27], Woschni [28] and Hohenberg [29]. The correlation proposed by Annand & Ma included a radiant term as well as a term including the angular velocity of the crank shaft. Woschni’s proposal was based on heat balance instead of surface wall temperatures and also included a variable for the mean gas velocity in the cylinder, as well as the mean piston speed. Hohenberg proposed a formulation similar to Woschni’s formulation, including the instantaneous cylinder volume. He also modified the exponent of the temperature term and slightly modified the gas velocity estimation. These three different semi-empirical correlations are shown in Appendix A.

Attempts have been made, by many authors, to contribute with new global correlations for the instantaneous heat transfer coefficient [30–33]. These correlations have usually been produced from experimental work validated to a specific engine or combustion type. Furthermore, there are different heat transfer correlations for in-cylinder heat transfer and heat transfer in other parts of the engine, like the intake or exhaust ports [34]. Sanli et al. [35], presented a summary of the available correlations for SI-engines, as well as an evaluation of these, showing that they do in fact differ quite a bit in their predictions of heat flux and heat transfer coefficients. Even though many authors have proposed their own global heat transfer models, the Woschni model is still the most famous and widely used model for diesel engines.

These global heat transfer models are most often used for heat transfer estimation in one-dimensional gas exchange codes, where the engine system is simulated, as exemplified by [36]. Another application area of these models is heat release analysis, where the Woschni model is dominating over other existing models.

### 2.2.2 Multidimensional models

More recently the use of multidimensional models, more specifically computational fluid dynamics (CFD) codes, for engine simulations has increased. In these multidimensional models the engine geometry is resolved in two or three dimensions, with a finite number of computational cells, either for steady state or transient calculations. In the early days of multidimensional modeling, the field was dominated by finite element calculations for temperature distribution and thermal stress estimation in the solid parts of the engine, as shown in [37–43]. More recently the use of finite volume based codes has allowed for simulations of the flow motions of the gases in the cylinder, along with the temperature distribution in the gases and the convective heat transfer to the solid parts [44, 45].

The main advantage with CFD simulations is that a successful simulation can provide values for any parameter at any location in the engine, whereas engine experiments are limited by accessibility. The simulations can even be carried out for extreme operating conditions, which is usually not an option for experimental work. CFD simulations in engines has, however, always been a difficult task because the level of complexity is relatively high. This applies both for the geometry, which not only contains moving parts but also contains areas that need extremely fine grid resolutions, and the physical processes that occur in the engine, such as turbulent
flow, fuel spray injection and combustion. Giving a fair representation of both the geometry and the physics in the engine will result in a large amount of cells along with extremely many time steps for the simulation. This is not often feasible for engine design work, because the time spent on computational mesh generation and simulation may become far too long. For this reason simplifications, both for the geometry and physics, are usually made to reduce the time spent on simulation work. This calls for the use of sub-models which should represent the physical behavior of the processes in the engine cylinder, such as spray behavior and combustion.

The development of CFD models for diesel engine application is an ongoing process. Reitz and Rutland [46] list a few of the models needed in order to successfully simulate a diesel engine cycle, as well as their development until 1995. Since then, there have been vast improvements, such as the formulation of temperature wall functions [47] and near wall treatment for turbulence in combination with conjugate heat transfer [48]. Spray and combustion models have also seen great progress in recent years.

In CFD studies, gas-exchange codes are frequently used to provide initial values and/or boundary conditions for the simulations. This is especially true for the calculation of the temperature distribution in solid components of the engine, where the heat transfer coefficient is one of the boundary conditions, such as in [49]. Other studies, such as [50] use the one-dimensional codes to obtain wall temperatures as boundary conditions, in order to simplify the computational domain. Along with more computational power, more and more researchers have included both the bulk gas and the solid regions in the computational domain, which means that the wall temperatures are resolved in the CFD simulation [44, 45, 51–53].

2.3 Current work

The research work carried out within the field of CFD in the automotive community has been quite focused on resolving the flow structures within the engine cylinder, fuel-air mixing and combustion modeling. The evaluation and documentation of energy transport, i.e., heat transfer, has been lacking and therefore it has been selected as the main objectives for the D60 project to further understand and document energy transport within the engine. When the heat transfer mechanisms within the engine are more clear, possible solutions can be proposed to reduce heat losses and by that increase the overall efficiency of the diesel engine.

In order to validate the simulation work, experimental work from the other project groups within D60 will be used along with one-dimensional engine system models provided by one of the project groups. A contribution from the CFD work is to formulate a global heat transfer model, which will be used to evaluate heat transfer in the one-dimensional system simulations.

There are, of course, limitations to the work carried out and presented in this thesis. The major limitation is that the computational domain only consists of an engine segment, which only simulates the closed volume part of the engine cycle. This means that the effects from the scavenging part of the cycle will be lost.
One other considerable limitation is the fact that the other project groups have not been able to run experiments on the engine selected for the project, due to technical problems. The experimental data, used for validation purposes in this work, have therefore been taken from other experiments performed on another similar Scania D13 engine. Some of the experimental data have no, or unclear, boundary conditions. Thus some of the boundary conditions had to be estimated.
A good CFD simulation is dependent on how well the computational domain and mathematical models reflect the actual process. In this chapter the most important models are discussed and those that have been chosen for this work are presented.

3.1 Turbulence model

It is of great importance to have a good representation of the flow field in the engine cylinder. The flow field solution is used by other models and will therefore affect the solution of those. There is a variety of available options to solve the flow field. The governing equations for the fluid flow are the conservation equations for mass and momentum, often referred to as the continuity and Navier-Stokes equations, respectively, see Appendix B. To solve this set of differential equations, the computational domain is discretized into a finite number of volumes and each volume element provides a local solution to the flow. Figure 3.1 shows how the energy cascade on different length scales in a turbulent flow. It shows how energy is increased by bulk motion on the largest flow scales, while energy decreased in the inertial range and dissipates into heat in the dissipation range.

3.1.1 Direct Numerical Simulation

Solving the governing equations directly, on all scales, for the flow in each volume element would be classified as a Direct Numerical Simulation or DNS. This requires immense computing resources because the smallest flow scales require the spatial grid resolution to be very high. This alternative is not a feasible option for the industry engine development, due to these requirements on computational power. It is, however, more often used for deeper understanding on small scale flow structures which can be used to further develop other turbulence models. The applicability of DNS is, for the time being, bound to low to moderate Reynolds numbers on small
computational domains [55]. This is why a part of the length scale spectrum, or even all of it, is modeled by making some approximations to the flow.

### 3.1.2 Large Eddy Simulation

When the largest length scales are resolved and the smallest scales are modeled, the simulation method is called *Large Eddy Simulation*, or *LES*. This method contains the instantaneous fluctuations in the large scale turbulent motion and can therefore give an accurate representation of the fluid flow, even though the smallest scales are modeled. With easier access to more computational power, this solution has become more attractive for industrial use, but it is still mainly used in the academic sector of the engine community.

### 3.1.3 RANS Simulation

In order to save computational power, and time, the option of modeling the whole length scale spectrum is taken. This is usually done by applying Reynolds decomposition on the governing differential equations, leading to the modeling group called *Reynolds Averaged Navier Stokes*, or *RANS* models. The RANS equations are shown in Appendix B. This method results in some loss of information, because only the time averaged values of the velocity components are available. Thus no instantaneous fluctuations are captured. RANS models are generally divided into groups, depending on the way of handling the closure problem introduced by the Reynolds decomposition. There are two main groups [55], *eddy viscosity based closure* and *Reynolds stress closure*, which are further divided into subgroups.
3. Model Set-up

Even though the Reynolds stress models (RSM) have shown improved results over the eddy viscosity based models (EVM), the difference in computational time and memory requirement have resulted in domination of EVM over RSM in the automotive community. The most widely used EVM is some form of the $k-\epsilon$ or the $k-\omega$ models, which are 2-equations EVM. These models are known to have difficulties to predict the flow correctly in some conditions present in engine simulations. Examples of these conditions are curved surfaces, secondary flows and rotational flows. In some cases, modifications have been made to these models to improve the prediction for such cases, but then usually the computational time in increased severely [56].

An alternative to the standard two equation models is to use non-linear EVM’s. The non-linear EVM’s include a rotational tensor that improves the handling of rotational- and secondary flows. The model used for the work presented in this thesis is a non-linear EVM, called $k-\zeta-f$, proposed by Hanjalic et al. [57]. It is based on the elliptic relaxation concept used in Durbin’s $v^2-f$ model [58], but introduces a velocity scale ratio, $\zeta = v^2/k$, instead of solving the equation for the velocity $v^2$. This simplifies the near wall behavior and improves numerical stability. Popovac & Hanjalic even proposed a compound wall treatment [59, 60], which is described in detail in Appendix B. This has been found to give improved results in engine applications.

3.2 Combustion model

While there is a constant development, as well as new additions, in the field of combustion modeling, only a handful of models are available in commercial CFD tools. The most widely used models readily available are the Eddy break-up model, the PDF model and the 3-Zones Extended Coherent Flame Model (ECFM-3Z). The PDF option has not been thoroughly studied in this work, so it will not be included in this review.

3.2.1 Eddy break-up model

The eddy break-up model is based on the assumption that the mean turbulent reaction rate is determined by the rate of dissipation of the turbulent eddies, containing fuel and oxygen [61]. The mean reaction rate of the fuel is given by Eq. (3.1).

$$\rho f_u = \frac{C_{fu}}{\tau_R} \rho \min \left( \frac{\overline{\nu_{fu}}}{S}, \frac{C_{Pr} \overline{\nu_{Pr}}}{1 + S} \right)$$  \hspace{1cm} (3.1)

This model has been used in commercial CFD tools to predict diesel combustion for a number of years and usually gives reasonable results. For a part of the work presented in this thesis, this model has been used for conventional diesel combustion.
3.2.2 ECFM-3Z model

The 3-zones extended coherent flame model (ECFM-3Z) was presented in 2004 by Colin & Benkenida [62]. This extension of the previously known Coherent Flame Model (CFM) and Extended Coherent Flame Model (ECFM), intended for SI combustion only, extended the use to diesel applications as well. This model should be capable of predicting the combustion for both cases. The addition to the ECFM model, is that a mixing model has been added to be able to handle both single and multiple fuel injections. Each computational cell is divided into three mixing zones, both for burnt gases and unburnt gases, resulting in a total of 6 zones in each computational cell. This model has been selected to be used for the PPC combustion mode in this work, due to its capability to handle both premixed and diffusion combustion.

3.3 Emission models

The most important emissions to keep track of in the combustion in large diesel engines are the four regulated emissions. According to the European Union emission regulation standards these are carbon monoxide (CO), hydrocarbons (HC), nitrogen oxides (NOx) and particulate matter (PM) [63]. Aside from these it is important to reduce the amount of CO2 emissions, but those have not been strictly regulated yet. The amount of CO2, CO and HC is predicted by the combustion model, but the prediction of NOx and soot requires additional modeling.

3.3.1 NOx modeling

Generally, nitric oxide, or NOx, formation stems from three main sources. These are named thermal NOx, prompt NOx and fuel NOx. NOx formation from prompt and fuel NOx are sometimes neglected in engine simulations [64] and therefore, only thermal NOx has been modeled in this work. The reaction mechanism for thermal NOx is usually expressed in terms of the so-called extended Zeldovich mechanism, given by Eqs. (3.2) to (3.4).

\[ \begin{align*}
N_2 + O & \xrightleftharpoons[k_{1b}]{k_{1f}} NO + N \\
N + O_2 & \xrightleftharpoons[k_{2b}]{k_{2f}} NO + O \\
N + OH & \xrightleftharpoons[k_{3b}]{k_{3f}} NO + H
\end{align*} \]

In Eqs. (3.2) to (3.4), the reaction rate constant \( k_{if} \) refers to the rate of the forward directed reaction \( i \). At the same time the reaction rate constant \( k_{ib} \) refers to the rate of the backwards directed reaction \( i \).

The model used in this work is based on the partial equilibrium assumption of the first two reactions of the extended Zeldovich mechanism. This leads to a global
reaction approach for the thermal nitric oxide formation, according to Eq. (3.5)

\[ \text{N}_2 + \text{O}_2 \rightarrow 2 \text{NO} \]  

(3.5)

With the reaction rate constant, \( k_f = k_{1f} \cdot k_{2f} \), the source term in the conservation equation for NO is given by Eq. (3.6).

\[ \frac{\partial c_{\text{NO}}}{\partial t} = 2 k_f c_{\text{N}_2} c_{\text{O}_2} \]  

(3.6)

### 3.3.2 Soot modeling

There are a few soot models available in the chosen commercial CFD code. The two most recently introduced are called the *Lund flamelet model* and the *Frolov kinetic model*. The former option considers contributions for the source term of the soot volume fraction, i.e., surface growth, oxidation, particle inception and fragmentation [65]. These four contributions can all be adjusted to the specific problem, by a scaling factor. This results in four scaling factors for the model.

The latter option is based on a detailed chemical reaction scheme for the calculation of soot formation and oxidation [64]. While the complete detailed kinetics scheme of the soot formation process consists of 1850 gas-phase reactions, 186 species and 100 heterogeneous reactions, the currently used model uses reduced number of species and reactions. This reduction is done in order to provide a computationally efficient kinetic soot model. In the CFD code, exact reactions constants have been incorporated for a few fuel types, but for other fuel types the solver automatically decides the most appropriate model constants. This means that no “tuning” is required by the user. This is the model used for the soot prediction in this work.

### 3.4 Spray behavior

In order to resolve the spray behavior, a set of sub-models is used. To account for droplet evaporation the *heat and mass transfer analogy*, or the *Dukowicz model*, has been applied. This model is based on the following assumptions: spherical symmetry, quasi steady gas-film around the droplet, uniform droplet temperature, uniform physical properties of the surrounding fluid and liquid - vapor thermal equilibrium on the droplet surface [66].

There are quite a few droplet break-up models available. They have not been extensively studied for the purpose of this work and therefore, one of them has been chosen by recommendation. This model is the *Wave child break-up model*, where a so-called blob injection is assumed, i.e., the largest droplet has the same size as the injection hole. This droplet is then divided into a few child droplets. There are six model parameters in the Wave child break-up model (\( C_1 - C_6 \)), two of which are fixed (\( C_1 \) and \( C_3 \)). One parameter (\( C_2 \)) estimates the characteristic break-up time, while parameters \( C_4 \) and \( C_5 \) control the size and number of child parcels. The distribution between the minimum stable diameter and the parent drop diameter is decided by the model parameter \( C_6 \).
3.5 Wall heat transfer

The enthalpy of the fluid, and thereby temperature as well, is solved in each computational cell with the energy equation, shown in Eq. (3.7).

\[
\rho \frac{DH}{Dt} = \rho \left( \frac{\partial H}{\partial t} + U_j \frac{\partial H}{\partial x_j} \right) = \rho \dot{q}_g + \frac{\partial P}{\partial t} + \frac{\partial}{\partial x_i} \left( U_j \tau_{ij} \right) + \frac{\partial}{\partial x_j} \left( \lambda \frac{\partial T}{\partial x_j} \right) \tag{3.7}
\]

In the equation above, the second term on the right hand side is a pressure-time derivative accounting for the contribution of local pressure rise to the energy equation. The third term on the right hand side includes the contribution from viscous heating to the energy equation. The remaining source term \(\dot{q}_g\) contains all remaining sources to the energy equation, such as chemically released heat from combustion and radiation. All of these are included in the simulations, but if any of them were to be excluded it could be the viscous heating term. This, however, would require a small numerical study in each case to ensure that it is not contributing to any energy sources.

The turbulence model used is a high Reynolds number based model, which requires special treatment of the near-wall region. This means that the heat transfer to the walls must be estimated by a special wall function. In the CFD code used for this thesis, there are two main options for the temperature wall function. The "standard" temperature wall function, proposed by Launder and Spalding, and the temperature wall function proposed by Han & Reitz. These are described in the following sections.

3.5.1 "Standard" temperature wall function

A widely used and presented temperature wall function is the one presented by Launder & Spalding [67] as the best practice of the Imperial College of London, at that time. In the CFD code used, it has been called the "standard" temperature wall function and is given by Eq. (3.8). It’s basis is that the first computational point is sufficiently remote from the wall so that the viscous effects are overwhelmed by the inertial ones.

\[
T^+ = \frac{\rho c_p \mu_r (T - T_w)}{q_w} = \sigma_{T,l} \left[ u^+ + P \left( \frac{\sigma_{T,l}}{\sigma_{T,l}} \right) \right] \tag{3.8}
\]

In the above equation, \(P\) is a correction function dependent on the ratio of laminar to turbulent Prandtl numbers, called the "pee-function".

3.5.2 Han & Reitz temperature wall function

In 1996, Han & Reitz proposed a wall temperature wall function for variable-density, turbulent flows in engines [47]. The formulation is based upon a few assumptions. The first assumption states that wall normal gradients are assumed to be much greater than tangential ones. The second assumption is that near-wall fluid velocity
is directed parallel to the wall. The third assumption states that pressure gradients are neglected. The fourth assumption declares that viscous dissipation is neglected. The fifth assumption is that radiation heat transfer is neglected and finally the gas is considered to be ideal. Basing the derivation of the temperature profile on the one-dimensional formulation of the energy equation, the resulting profile is given by Eq. (3.9).

\[ T^+ = 2.1 \ln(y^+) + 2.1G^+ y^+ + 33.4G^+ + 2.5 \]  

where \( G^+ \) is a dimensionless source term in the energy equation, given by Eq. (3.10).

\[ G^+ = \frac{G \nu}{q_w u^*}, \quad G = \overline{Q}_c \]  

where \( \overline{Q}_c \) is the average chemical heat release. The corresponding wall flux formulation is given by Eq. (3.11).

\[ q_w = \frac{\rho c_p u^+ T \ln(T/T_w) - (2.1y^+ + 33.4)G \nu / u^+}{2.1 \ln(y^+) + 2.5} \]  

If the source term \( G \) can be neglected, Eq. (3.11) will reduce to Eq. (3.12).

\[ q_w = \frac{\rho c_p u^+ T \ln(T/T_w)}{T^+} \]  

Nuutinen et al. [48] examined the formulation proposed by Han & Reitz and found that their formulation neglects the fact that the friction velocity \( u_\tau \) should be a function of a variable density. Han & Reitz, however, had fixed their friction velocity to a certain density and then treated the density as a variable, treating the friction velocity as a constant. Nuutinen et al. then suggested an alternative choice of dimensionless parameters, keeping the remaining Han & Reitz formulation, with acceptable results. This altered formulation is, however, not available in the currently used CFD code.

For the purpose of this work, the "standard" temperature wall function of Launder & Spalding has been used. The impact of different temperature wall functions is an interesting option for future work.

It is clear that the use of wall functions is an approximation and the physical behavior within the boundary layer is not fully understood. Robinson [68] demonstrated that small scale "hairpin-shaped" vortical structures disrupt the boundary layer and cause the previously believed laminar behavior of the boundary layer to be somewhat false. These findings have also been demonstrated by a DNS simulation by Wu & Moin [69]. If this is a frequent behavior in the near-wall region, then the assumption made in the derivation of the wall functions, that the wall normal gradients are dominant over other gradients, might be false.
Chapter 4

Studied Cases

This chapter contains the description of the engine used for simulations, description of the baseline cases along with a short description of the studies performed for this thesis work.

4.1 Engine geometry

The simulated engine is a 13 l, six cylinder diesel engine, with the engine and injector specifications for a single cylinder given in Table 4.1. To simplify the computational domain, a closed volume mesh was built enclosing one out of the eight sprays from the injector, resulting in a 45° slice of the engine cylinder. On each side of the spray, a cyclic boundary condition was applied. The term ‘closed volume’ indicates that only the compression, combustion and expansion part of the engine cycle is simulated, neglecting the gas-exchange part of the engine cycle.

The average cell size in the computational mesh ranged from 0.6 mm to 1.2 mm and there were two cell layers placed adjacent to the wall boundary of 0.1 mm thickness.

<table>
<thead>
<tr>
<th>Table 4.1: Scania D13 engine geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement [l]</td>
</tr>
<tr>
<td>Geom. Comp. Ratio [-]</td>
</tr>
<tr>
<td>Conn. Rod Length [mm]</td>
</tr>
<tr>
<td>Bore x Stroke [mm]</td>
</tr>
<tr>
<td>Valve-train</td>
</tr>
<tr>
<td>IVC = 40 CAD ABDC</td>
</tr>
<tr>
<td>EVO = 50 CAD BBDC</td>
</tr>
<tr>
<td>Spray angle [deg]</td>
</tr>
<tr>
<td>Orifices [-]</td>
</tr>
<tr>
<td>Orifice diameter [mm]</td>
</tr>
</tbody>
</table>
4. Studied Cases

The mesh was divided into 17 angular cells, resulting in mesh densities of around 113,000 cells at BDC and 66,000 cells at TDC. The mesh is shown at the position 20 crank angle degrees before top dead center in Fig. 4.1.

Two separate studies were performed on the engine, a parameter study and a heat rejection study, for two different combustion modes.

4.1.1 Boundary & initial conditions

As always, when performing a CFD simulation, a set of boundary conditions is needed to close the system of equations. The boundaries can either have a fixed temperature or a fixed heat flux as a boundary condition. For the baseline case, the wall boundaries were fixed to a certain temperature, obtained from a gas-exchange simulation. The wall boundaries in the simulations include the engine piston, cylinder head and liner, and the temperatures used are 570 K, 560 K and 430 K, respectively. The baseline case, chosen for this work, was a full load, diesel combustion case provided by the engine manufacturer. The operating conditions are provided in Table 4.2.

4.2 Parameter study

For the purpose of this thesis, a parameter study was performed on the modeled engine according to the methodology shown in an earlier publication [70]. The aim of this study is to investigate the effect of certain parameters on the wall heat transfer and, to some extents, the performance of the engine. After validating the baseline case with an experimental set-up, three parameters were chosen for the study. The parameters chosen for this investigation were exhaust gas recirculation (EGR), injection duration and swirl number. Table 4.3 shows the chosen parameters along with their values in the study.

The simulation set-up resulted in a $6 + 1 = 7$ simulation runs, as it contains one baseline case and six variations to that set-up.
Table 4.2: Operating conditions for the diesel baseline case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Effective compression ratio</td>
<td>16.8 [-]</td>
</tr>
<tr>
<td>Engine speed</td>
<td>1250 [RPM]</td>
</tr>
<tr>
<td>Swirl ratio</td>
<td>1.7 [-]</td>
</tr>
<tr>
<td>Inlet pressure @IVC</td>
<td>4.15 [bar]</td>
</tr>
<tr>
<td>Inlet temperature @IVC</td>
<td>350 [K]</td>
</tr>
<tr>
<td>Residual gas</td>
<td>28.1 [%]</td>
</tr>
<tr>
<td>Start of injection</td>
<td>2.6 [CAD BTDC]</td>
</tr>
<tr>
<td>End of injection</td>
<td>57.4 [CAD ATDC]</td>
</tr>
<tr>
<td>Fuel mass</td>
<td>310 [mg/cyl]</td>
</tr>
<tr>
<td>Fuel type</td>
<td>Diesel MK1 [-]</td>
</tr>
</tbody>
</table>

Table 4.3: Values for the parameter study

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Lower value</th>
<th>Baseline</th>
<th>Higher value</th>
</tr>
</thead>
<tbody>
<tr>
<td>EGR [%]</td>
<td>18.1</td>
<td>28.1</td>
<td>38.1</td>
</tr>
<tr>
<td>Injection duration [CAD]</td>
<td>40</td>
<td>60</td>
<td>80</td>
</tr>
<tr>
<td>Swirl number [rpm]</td>
<td>1.0</td>
<td>1.7</td>
<td>2.4</td>
</tr>
</tbody>
</table>

4.3 Heat rejection study

After the simulation cases had been validated with experimental cases, a study was performed to examine how different heat rejection levels affected the engine performance and emission levels. This was done for two combustion modes, a conventional diesel combustion mode and the newly introduced PPC combustion mode. The validated cases were subjected to four different heat flux boundary conditions. In stead of a fixed temperature, a boundary condition of a fixed heat flux through the wall boundary was assigned. The computed heat flux from the baseline case was scaled to give 75%, 50%, 25% and 0% heat flux, which were then subjected to each wall in four different simulation cases. The engine response, in terms of performance and emissions, was compared for these cases.

The baseline case for the diesel combustion, used in this part of the study, was the full load case described in Table 4.2.

The PPC combustion case used is a 20.8 bar IMEPg load case from the experiments performed by Manente [21]. The operating conditions for that case are given in Table 4.4. It is worth noting that while the engine parameters are the same as in other parts of the study, the effective compression ratio has been reduced from 16.8 to the value 15.5. This was done in the calibration process of the CFD model, and might be explained by the conditions of the engine cylinder when the experiments were performed. Losses like leakage and blow by are not accounted for in the CFD model and these might have been excessive during the experimental
Table 4.4: Operating conditions for the PPC baseline case

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Effective compression ratio</td>
<td>15.5 [-]</td>
</tr>
<tr>
<td>Engine speed</td>
<td>1250 [RPM]</td>
</tr>
<tr>
<td>Swirl ratio</td>
<td>2.095</td>
</tr>
<tr>
<td>Inlet pressure</td>
<td>3.66 [bar]</td>
</tr>
<tr>
<td>Inlet temperature</td>
<td>309 [K]</td>
</tr>
<tr>
<td>EGR</td>
<td>55.24 [%]</td>
</tr>
<tr>
<td>Start of injection</td>
<td>4 [CAD BTDC]</td>
</tr>
<tr>
<td>End of injection</td>
<td>22 [CAD ATDC]</td>
</tr>
<tr>
<td>Fuel mass</td>
<td>195 [mg/cyl]</td>
</tr>
<tr>
<td>Fuel type</td>
<td>Diesel MK1 [-]</td>
</tr>
</tbody>
</table>

procedure. Furthermore, the CFD code simulates a rigid connecting rod, which is not physically accurate and will affect the compression ratio. The swirl ratio has also been increased from the conventional 1.7 to 2.095. The residual gas amount is roughly doubled, i.e., it is increased from 28.1% to 55.24% and the injection duration is reduced by more than half the time compared to the diesel case. The inlet pressure and temperature are reduced for the PPC combustion mode, compared to the diesel case.
Chapter 5

Results & Discussions

This chapter starts with a presentation of the results from the validation process for both combustion modes, diesel and PPC. Then results from the respective studies are presented.

5.1 Baseline validation

There are two baseline cases used for this work, one conventional diesel combustion case and one partially premixed combustion case, as described earlier.

5.1.1 Diesel combustion

The diesel combustion case, as stated earlier, was a full load case with the gross indicated mean effective pressure (IMEP) of 25.6 bar. Figure 5.1 shows the mean pressure trace for the simulated engine compared to the experimental data and as this figure shows, the CFD results provide an acceptable fit to the experimental results.

Figure 5.2 shows the heat release characteristics for both experimental values and values from CFD simulations. The left hand figure shows the accumulated heat release as a function of crank angle degree, whereas the right hand figure shows the heat release rate as a function of crank angle degree. The values for the accumulated heat release are not identical for the experimental data and the CFD simulations. They differ about 2-3 kJ in total. This is due to the fact that the CFD code provides the chemical heat release from combustion, while the experimental data provided the apparent heat release, based on the measured pressure trace. The effects of evaporation and heat losses are visible in the experimental data, while the chemical heat release contains no heat transfer effects. The trends in the experimental heat release rate are, however, well captured by the CFD simulation.
5. Results & Discussions

![Figure 5.1: Mean Pressure trace from experiments and CFD simulations.](image1)

![Figure 5.2: Accumulated heat release and heat release rate from experiments and CFD simulations.](image2)

When studying both the pressure trace and the heat release rate, one can note that the start of combustion is slightly earlier for the CFD results. This can either be a result of too fast evaporation of the spray droplets in the model or lower wall temperatures in the model compared to the experimental set-up. This leads to a slower combustion, yielding lower peak pressure and a longer tail in the rate of heat release shape.

### 5.1.2 Partially premixed combustion

As stated before, the PPC combustion case was a high load case from the experiments of Manente et al. [21], with the gross indicated mean effective pressure of 20.8 bar. The mean pressure trace is shown in Fig. 5.3 and it shows that the CFD model does very well in predicting the pressure during compression and the first stage of combustion. However, when the combustion slows down the CFD simulations over-predict the speed of combustion, leading to a certain phase shift in the pressure trace. This has proven to be extremely difficult to capture with the existing combustion model, but the fit presented is quite acceptable.
The heat release characteristics are shown in Fig. 5.4. The left hand figure shows the accumulated heat release, while the right hand figure shows the heat release rate. The heat release from the experiments in this case have been adjusted to show the chemical heat release, so the effects of heat transfer have been added to the total heat release. The total amount of heat released is almost the same for the simulations and experiments, while there seems to be a phase shift in the heat release rate shape as was also noted in the pressure trace. There is, however, a spike in the heat release rate in the beginning, which is probably due to too fast evaporation of the fuel droplets.

It is worth noting that the load point selected for the ppc case was different to the one for the diesel case. The ppc load case was a 75% load, while the diesel load point was a full load case. It should also be noted that the effective compression ratio was reduced from 16.8 to 15.5. Comparing the heat release from the ppc case to the diesel case, it is evident that the amount of heat released is considerably less in the ppc case, which is both due to the difference in combustion modes and also the difference in load points.
5. Results & Discussions

5.2 Parameter study

The results of the parameter study are presented below, both graphically and in a Table. For an estimation of the indicated engine output, the mean pressure was used to evaluate the indicated mean effective pressure during compression and expansion, or IMEP\textsuperscript{co}. Because the simulation only considers the closed volume cycle, the resulting IMEP\textsuperscript{co} is not directly comparable to the traditional IMEP\textsubscript{g}. Table 5.1 contains the evaluated values of the IMEP\textsuperscript{co} for all parameters. Changes in mean cylinder pressure and temperature, along with heat release rate and wall heat flux is examined graphically.

### Table 5.1: Values for IMEP\textsuperscript{co} (bar) for the parameter study

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Lower value</th>
<th>Baseline</th>
<th>Higher value</th>
</tr>
</thead>
<tbody>
<tr>
<td>EGR</td>
<td>27.1</td>
<td>26.7</td>
<td>26.2</td>
</tr>
<tr>
<td>Injection duration</td>
<td>29.0</td>
<td>26.7</td>
<td>23.5</td>
</tr>
<tr>
<td>Swirl number</td>
<td>26.4</td>
<td>26.7</td>
<td>26.5</td>
</tr>
</tbody>
</table>

5.2.1 EGR variation

The amount of exhaust gas recirculated was varied by 10%, around the baseline case, as shown in Table 4.3. By reducing the EGR amount by 10% resulted in roughly a 0.5 bar increase in the IMEP\textsuperscript{co} and the increase of the EGR amount, by 10% resulted in an equally large decrease of the IMEP\textsuperscript{co}.

![Figure 5.5: Mean pressure of the EGR variation.](image)

Figures 5.5 to 5.7 show a set of graphical results from the simulations. In Fig. 5.5, the mean pressure trace is shown as a function of crank angle degree. There is no clear difference between the different EGR cases in the compression stroke, while the pressure during combustion is slightly different. This difference is also visible.
in Fig. 5.6b, which shows the crank angle dependent mean temperature. These differences are mainly due to different results from the combustion, as shown by the heat release rate from combustion in Fig. 5.6a. Between 10 and 25 CAD ATDC, a difference in the chemical heat release is clear between the EGR cases. This is due to the fact that exhaust gas serves as a "buffer" in the combustion process, providing cooling effects on the flame temperature. This is why a larger amount of EGR gives both lower temperatures and lower IMEP\textsuperscript{co}.

By affecting the in-cylinder pressure and temperature, changing the amount of EGR also affected the wall heat transfer, as shown in Fig. 5.7. The largest effect is shown in the heat flux through the piston surface, while cylinder head and liner are less affected. This is because most of the combustion takes place near the piston bowl area, but further from the cylinder liner and head. The peak heat flux to the piston surface occurs between 10 and 30 CAD ATDC and the maximum difference between the EGR cases also occurs there, which corresponds to the timings for the maximum changes in pressure and temperature.
5. Results & Discussions

5.2.2 Injection duration variation

The injection duration was varied by 20 crank angle degrees around the baseline case, with the value of 60 CAD. As Table 5.1 shows, by reducing the injection duration by 20 CAD results in an increase in IMEP\textsubscript{co} of 2.3 bar, while the increase of the injection duration by 20 CAD results in a decrease in IMEP\textsubscript{co} by over 3 bar.

\begin{figure}[h]
\begin{center}
\includegraphics[width=0.8\textwidth]{figure5_8.png}
\end{center}
\caption{Mean pressure of the injection duration variation.}
\end{figure}

Figure 5.8 shows the mean cylinder pressure for the different injection duration cases. From the figure it is clear that reducing the injection duration to 40 CAD affects the peak pressure, resulting in higher power output of the engine, which provides higher efficiency. This, however, is accompanied by higher mechanical losses, such as friction, and more noise from higher pressure derivatives. The increased injection duration, to 80 CAD, shows severely reduced peak pressures, leading to near-misfire, and thereby lowered efficiency of the engine.

\begin{figure}[h]
\begin{center}
\includegraphics[width=0.8\textwidth]{figure5_9.png}
\end{center}
\caption{(a) Heat release rate \hspace{1cm} (b) Mean temperature}
\end{figure}

Figure 5.9: Mean temperature and heat release rates for the injection duration parameter sweep.
The temperature and the heat release rate are shown in Fig. 5.9a. They show that combustion does occur in the case of the 80 CAD injection duration. The shapes of the heat release rate are quite different between the cases, where both combustion start and duration varies with the different injection duration. The short duration results in a quicker ignition and a shorter combustion duration, which gives higher peak temperatures.

This is also clearly visible in the wall heat fluxes, shown in Fig. 5.10, where all surfaces experience large variations in heat flux between the different injection duration cases. By decreasing the injection duration from 60 to 40 CAD ATDC results in change in combustion behavior, leading to a faster combustion and higher bulk gas temperatures. This results in a change of factor two in the peak wall heat flux. The change in the opposite direction, from 60 to 80 CAD injection duration, leads to a reduction in peak wall heat fluxes of around factor two. This applies for both piston and cylinder head. However, for the cylinder liner the effects are opposite. Increasing the injection duration from 60 to 80 CAD results in an increase in peak heat flux, while the decreasing injection duration lowers the peak heat flux. This might be the result of a slower combustion for the increased injection duration, i.e., for the slower combustion the evaporation of fuel droplets might be slower and the spray might penetrate further resulting in a combustion that is closer to the cylinder liner. This, of course, would lead to higher local temperatures near the liner and, therefore, higher wall heat fluxes.

Figure 5.10: Wall heat fluxes for the injection duration variation.
5. Results & Discussions

5.2.3 Swirl number variation

The swirl number, defining the rotational movement of the in-cylinder gas, was varied by 0.7 around the baseline value of 1.7. This variation resulted in marginal effects on the IMEP$^{\text{co}}$ and furthermore, there were no clear trends in the results. The reduction of the swirl number by 0.7 gave a decrease of 0.2 bar in IMEP$^{\text{co}}$, while the increase of the swirl number by 0.7 also resulted in a decrease of the IMEP$^{\text{co}}$, now by only 0.1 bar.

![Figure 5.11: Mean pressure of the swirl number variation.](image)

The mean pressure trace is shown in Fig. 5.11. There are very slight differences in the pressure during combustion, where the lower swirl number gave lower peak pressure. It might be that the higher swirling flow provides a better stratification of the combustion, leading to a slightly more efficient combustion.

![Figure 5.12: Mean temperature and heat release rates for the swirl number parameter sweep.](image)

This is, however, neither visible in the heat release rate, nor the mean temperature, shown in Figs. 5.12a and 5.12b, respectively. In those figures, there are no clear
trends or variations. There are mild temperature variations between around 20 and 40 CAD ATDC.

![Figure 5.13: Wall heat fluxes for the swirl number variation.](image)

There are, however, noticeable differences in wall heat flux, especially for the piston and liner surfaces, as shown in Fig. 5.13. The reason for this difference might be that the increased swirling flow can have a cooling effect on the liner surface, while at the same time not affecting the cylinder head surface as much. It might also affect how the flow structures look like in the bowl area, leading to a reduced heat flux through the piston bowl. This parameter is, however, unlikely to have any dramatic effects on engine efficiency by itself.
5.3 Heat rejection study

The validated cases, for both diesel and PPC modes, were used in a heat rejection study, as described earlier. The results from the study are presented in the following sections for each combustion mode separately.

5.3.1 Diesel combustion

The heat release from combustion was examined as a function of crank angle degree for the different heat rejection levels, as shown in Fig. 5.14. It was found that reducing the heat rejection did in fact not affect the heat release shape as much as expected. Aronsson et al. [71] showed that higher wall temperatures should affect the combustion phasing and thereby the heat release rate shape. The change in combustion phasing leads to a more powerful premixed part of the flame and a slower diffusion flame, prolonging the combustion. This was not observed in the simulations performed for this thesis, so it should be a subject of a further study.

![Figure 5.14: Heat release from combustion for different heat rejection levels.](image)

The performance of the engine, in terms of indicated fuel consumption (ISFC), indicated efficiency and indicated mean effective pressure (IMEP), is shown as a function of the heat flux percentage in Fig. 5.15. There, 100% heat flux represents the validated case, while 0% presents an adiabatic case. As expected, and shown in a previous research [8], the indicated efficiency and power output of the engine is increased by introducing insulation effects on wall boundaries and the specific fuel consumption is reduced. The question remains if this behavior is at the cost of elevated peak temperatures and thereby increased emissions.
The mass fractions of NO\textsubscript{x} and soot are shown in Fig. 5.16 as a function of crank angle degrees for different heat rejection levels. Based on the figure, it seems that decreased heat rejection leads to increased temperatures and an increase in NO\textsubscript{x} production. However, the amount of soot in the cylinder after combustion is relatively unaffected by the decreased heat rejection, even though the peak in soot production is higher for lower heat rejection.

The average temperature in the cylinder, as a function of crank angle, for different heat rejection levels is given in Fig. 5.17. The figure does not show temperatures above 2000 K, which is often referred to as the the limit when thermal NO\textsubscript{x} begins to form, so it is likely that the NO\textsubscript{x} is formed due to some local temperature peaks.
The equivalence ratio, $\phi$, and the cylinder gas temperature for the baseline case are shown at a mid-plane of the engine sector in Figs. 5.18 and 5.19, respectively.

As shown by Akihama et al. [23], the production of NO\textsubscript{x} in diesel engines begins to occur at temperatures around 2000 K and equivalence ratios $\phi < 2$. Soot, on the other hand, is formed at temperatures between 1500 K and 2400 K for $\phi > 2$. This range of temperature and equivalence ratios only explains the formation of NO\textsubscript{x} and soot. Andersson et al. [72] stated that soot is oxidized at lower equivalence ratios, which introduces a trade-off between NO\textsubscript{x} and soot. The red color in the temperature field of the mid-plane section corresponds to temperatures above 2200 K and the red color for the equivalence ratio corresponds to $\phi = 2$ and higher. This shows that the soot is formed around the spray path between 10 CAD ATDC and 20 CAD ATDC and oxidized at later crank angle degrees. NO\textsubscript{x}, which is formed
at high temperatures and low equivalence ratios, is mainly formed after 20 CAD ATDC. This corresponds to the 2D emission plot given in Fig. 5.16.
5.3.2 Partially premixed combustion

The heat release was examined as a function of heat rejection level, as shown in Fig. 5.20, and it was found that no significant changes were made to the heat release with increasing insulation. This, again, implies that the combustion is not severely affected during this procedure and should be studied further.

![Figure 5.20: Heat release from combustion for different heat release levels.](image)

The performance of the engine, in terms of ISFC, indicated efficiency and IMEP, is shown as a function of insulation effects in Fig. 5.21. As for the diesel engine, the indicated efficiency and power output of the engine is increased by introducing insulation effects on wall boundaries and the specific fuel consumption is reduced. Compared to the diesel engine, the indicated efficiency is higher for the PPC case, leading to lower values of the specific fuel consumption.

![Figure 5.21: Engine performance for different levels of heat rejection.](image)

The mass fractions of NO\textsubscript{x} and soot are shown in Fig. 5.22 as a function of crank angle degrees for different heat rejection levels. Again, it seems that decreased heat rejection leads to increased temperatures and more NO\textsubscript{x} is produced. Unlike for
the diesel engine, the soot level at the end of simulation is not unaffected by change in insulation, i.e., decreasing heat rejection results in more soot in the cylinder. However, it is apparent that the amount of emissions for the PPC combustion is far less than in the diesel combustion, around three orders of magnitude less.

![Emission mass fractions for different levels of heat rejection.](image)

**Figure 5.22:** Emission mass fractions for different levels of heat rejection.

The average temperature in the cylinder, as a function of crank angle and for different heat rejection levels, is given in Fig. 5.23. It is clear that the PPC combustion results in lower average cylinder temperatures than in diesel combustion. This directly corresponds to reduced amounts of emissions, because neither soot nor NO\textsubscript{x} is formed at temperatures lower than around 1500 K. There are, however, zones with higher local temperatures and equivalence ratios in the cylinder that contribute to the amount of emissions actually generated.

![Average temperature in the cylinder for different heat rejection levels.](image)

**Figure 5.23:** average temperature in the cylinder for different heat rejection levels.

The equivalence ratio, \( \phi \), and the cylinder gas temperature for the baseline case are shown at a mid-plane of the engine sector in Figs. 5.24 and 5.25, respectively. The upper limit (red color) of the contour plots has the same value as for the diesel case, i.e., \( T = 2200 \) K and \( \phi = 2 \).
Figure 5.24: Equivalence ratio for the PPC combustion, from TDC to 50 CAD ATDC.

Figure 5.25: Cylinder gas temperature for the PPC combustion, from TDC to 50 CAD aTDC.

It can be seen that the temperature does not reach a high enough value for NO\textsubscript{x} to be formed in the plane chosen and the areas where the equivalence ratio is high enough for soot to be formed are almost all too cold for the soot production to start. This is in agreement with the emissions shown in Fig. 5.22.
Chapter 6

Conclusions and Future Work

Further knowledge of the temperature distribution inside an engine cylinder as well as the heat transfer process is critical in order to produce engines with higher efficiency, as well as lower emissions. The theoretical efficiency of the diesel thermodynamic cycle is proportional to the ratio of the peak temperature and the ambient inlet temperature in the system. This poses requirements on the material side which have no solutions, so in order to raise the efficiency of the engine, a detailed knowledge of the thermodynamic behavior of the system is required. This work is one step on the way towards deeper understanding of this phenomenon, and the parameter study included in this work highlights briefly how some parameters can affect the in-cylinder heat transfer. For a change in a single parameter, the injection characteristics showed more potential for efficiency increase than the other parameters examined. The combination of changes was, however, not examined and it would be interesting to investigate how a combination of parameters will effect the efficiency of the engine. The study could even be extended further by including more parameters and alternative sub-models.

A study of engine performance, along with an estimation of NO\textsubscript{x} and soot emission levels, was performed on a heavy duty diesel engine for two different combustion modes. A specific load point was used as a validation point for each combustion mode and experimental results were used for the validation process. Both baseline cases gave acceptable agreement with the experimental data.

The validated cases were used in a heat rejection study, where the resulting heat flux to the solid walls from the baseline case was scaled and used as wall boundary conditions for the remaining cases in the study. For the simulated cases, the shape of the heat release rate did not seem to be affected by the change in heat rejection, indicating that the combustion itself was relatively unaffected. This is in contradiction to results from experimental tests reviewed in this thesis and should therefore be a subject of further investigation.

Engine performance followed expected patterns for both combustion modes as a function of heat rejection, i.e., efficiency increased with reduced heat rejection as
well as BSFC reduced and IMEP increased. Decreased heat rejection caused an increase in NOx emissions in both cases, but only slightly increased soot emissions in the PPC case. There was, however, a difference of three orders of magnitude in the mass fractions of the emission between the diesel and PPC combustion mode. This is due to an appropriate combination of $\phi$ and $T$ in the PPC combustion mode.

The results from this thesis work show that the combination of low heat rejection and low temperature combustion is beneficial and will be a subject of further studies throughout the doctoral thesis work. Given that the aim of the study is to increase efficiency, at the same time maintaining within the regulated emission levels, investigation of various coating alternatives as well as alternative fuels will also be included.

After performing these closed volume engine segment simulations, the next step will be performing simulations for a full cylinder, coupled with the solid components and water cooling system. Those simulations would include the full engine cycle, also accounting for the gas-exchange part of the cycle. An important part of the engine heat transfer process is the solid heat conduction and coolant flow, which will also be evaluated. This will require the use of a finite element solver for the solid parts, while the water cooling will be solved using the same solver as for the gas flow.
Bibliography


[26] Eichelberg, G. “Some new investigations on old combustion engine problems”. 

[27] Annand, W., and Ma, T. “Instantaneous Heat Transfer Rates to the Cylinder 
Head Surface of a Small Compression-Ignition Engine”. Proceedings of the 

[28] Woschni, G. “A universally applicable equation for the instantaneous heat 
transfer coefficient in the internal combustion engine”. SAE International(64), 


“Evaluation of heat transfer correlations for HCCI engine modeling”. Applied 

[32] Chang, J., Güralp, O., Filipi, Z., Assanis, D., Kuo, T. W., Najt, P., and 
Rask, R. “New Heat Transfer Correlation for an HCCI Engine Derived from 
Measurements of Instantaneous Surface Heat Flux”. SAE paper 2004-01-2996, 
2004.

[33] Galindo, J., Luján, J., Serrano, J., Dolz, V., and Guilain, S. “Description of 
a heat transfer model suitable to calculate transient processes of turbocharged 
diesel engines with one-dimensional gas-dynamic codes”. Applied 

[34] Depcik, C., and Assanis, D. “A Universal Heat Transfer Correlation for Intake 
and Exhaust Flows in a Spark-Ignition Internal Combustion Engine”. SAE 

Evaluation by Models of Load and Spark Timing Effects on the In-Cylinder 

[36] Xin, Q. “Theoretical Analysis of Internal Combustion Engine Miscellaneous 


[38] Singh, V., Upadhyay, P., and Samria, N. “Some Heat Transfer Studies on 
a Diesel Engine Piston”. International Journal of Heat and Mass Transfer, 

Piston With and Without Ceramic Insulation on Crown Face”. International 
BIBLIOGRAPHY


Appendix A

Correlations for Global Models

In this appendix, the most widely semi-empirical global heat transfer correlations are presented. These are often used in one-dimensional gas-exchange codes for heat transfer estimation and heat release analysis. The basis for these correlations is a general correlation for convective heat transfer relating the Nusselt number to the Reynolds number as

\[ Nu = CRe^n \]  

(A.1)

A.1 Annand & Ma

After reviewing existing formulation at the time as well as available heat-transfer data, Annand proposed the following formulation for the global instantaneous heat flux in an engine cylinder.

\[ q'' = akDRe^{0.7}(T - T_w) + b(T^4 - T_w^4) \]  

(A.2)

Annand’s formulation was later extended, by Annand and Ma, by including the angular velocity of the crank shaft and the Stefan-Boltzmann constant, as shown in Eq. (A.3).

\[ q'' = kDRe^{0.7}\left[ a(T - T_w) + \frac{a'dT}{\omega} \right] + b\sigma(T^4 - T_w^4) \]  

(A.3)

The heat-transfer coefficient from the formulation in Eq. (A.3) is given in Eq. (A.4).

\[ h_{Annand \& \ Ma} = \frac{k}{B}Re^{0.7}\left( 0.12 + \frac{0.12}{\omega(T - T_w)} \right) \]  

(A.4)

A.2 Woschni

The Woschni model is also based on the similarity law of turbulent heat transfer. Instead of using the wall temperature, he decided to use a heat balance to determine
the total heat transferred to the combustion chamber walls. Woschni’s formulation is given by Eq. (A.5).

\[ h_{\text{Woschni}} = \alpha_s B^{m-1} p^{m T^{0.75-1.62m}} \left[ C_1 S_p + C_2 \frac{V_d T_1}{p_1 V_1} (p - p_0) \right]^m \] (A.5)

In Woschni’s formulation, the value of 0.8 is used for the exponent \( m \). The constant \( \alpha_s \) is a scaling constant, used to tune the heat transfer coefficient to match a specific engine geometry. A common value of the scaling constant is, as given in Heywood [5], \( \alpha_s = 3.26 \). This value of the scaling constant is used when the pressure unit is kPa, temperature unit is Kelvin, the characteristic length unit is meters, the volume unit is cubic meters and the mean piston velocity unit is meters per second. The values for the constants \( C_1 \) and \( C_2 \) is as follows:

- Gas exchange period: \( C_1 = 6.18 \) \( C_2 = 0 \)
- Compression period: \( C_1 = 2.28 \) \( C_2 = 0 \)
- Combustion and expansion period: \( C_2 = 3.24 \times 10^{-3} \)

### A.3 Hohenberg

After examining Woschni’s formulation, Hohenberg proposed a modified formulation. With Hohenberg’s formulation, the time-averaged heat flux prediction has been improved. Hohenberg included the instantaneous volume in his formulation and modified the velocity term, as shown in Eq. (A.6).

\[ h_{\text{Hohenberg}} = \alpha_s V^{-0.06} p^{0.8} T^{-0.4} (S_p + \beta)^{0.8} \] (A.6)

Hohenberg used the values of 130 and 1.4 for \( \alpha_s \) and \( \beta \), respectively.

### A.4 Chang

A research group from the University of Michigan and the GM Research and Development Center studied the heat transfer in a low temperature combustion engine, more specifically a HCCI engine. By performing and analyzing instantaneous surface temperature and heat flux measurements, a global heat transfer model was suggested in a publication by Chang et al. [32]. Using the Woschni formulation as a reference point, they produced a correlation on the following form.

\[ h_{\text{global}} = \alpha_{\text{scaling}} L(t)^{m-1} \frac{k}{\mu_m} p(t)^m T(t)^{-m} v(t)^m \] (A.7)

Following Woschni’s example, their approach was to tune the existing Woschni model for better agreement with the HCCI combustion mode. Their resulting formulation is shown in Eqs. (A.8) and (A.9).

\[ h_{\text{Chang}}(t) = \alpha_{\text{scaling}} L(t)^{-0.2} p(t)^{0.8} T(t)^{-0.73} v_{\text{tuned}}(t)^{0.8} \] (A.8)

\[ v_{\text{tuned}}(t) = C_1 S_p + \frac{C_2 V_d T_1}{6 \bar{p}_c \bar{V}_R} (p - p_{\text{mot}}) \] (A.9)
The Chang model has three main differences from the Woschni model. The characteristic length was changed from being the cylinder bore to the instantaneous chamber height. The temperature exponent is changed to -0.73, as opposed to the original value of -0.546. The $C_2$ constant is reduced to be $1/6$ of its original value. The Chang model was proposed based on measurements and analysis from one engine. This model would benefit from being verified using several engines of different sizes to ensure that the use of the instantaneous chamber height will give the right representation of the characteristic length in all cases.
Appendix B

Important Governing Equations

Performing CFD simulations on physical phenomena requires a mathematical representation of the physical processes in the system.

B.1 Flow equations

Flow with a viscous heat conducting fluid is governed by the momentum equations, also named Navier-Stokes equations, supplemented by the continuity equation and the energy equation. Sometimes, but incorrectly, the term Navier-Stokes equations is applied for the whole set of governing equations, i.e., momentum, continuity and energy. Equations (B.1) to (B.3) show the whole set of the governing equations, including continuity, Eq. (B.1), momentum, Eq. (B.2) and energy, Eq. (B.3).

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0 \tag{B.1}
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p \delta_{ij}}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{B.2}
\]

\[
\frac{\partial \rho e_0}{\partial t} + \frac{\partial \rho u_j e_0}{\partial x_j} = -\frac{\partial u_j p}{\partial x_j} + \frac{\partial u_i \tau_{ij}}{\partial x_j} - \frac{\partial q_j}{\partial x_j} \tag{B.3}
\]

The viscous stress is given by Eq. (B.4).

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \tag{B.4}
\]
B.2 Turbulence model equations

The model used in this thesis work is a non-linear eddy viscosity model, $k - \zeta - f$. The model is based on a set of equations, given below. The turbulent viscosity is given by Eq. (B.5) and the transport equation for $\zeta$ is given by Eq. (B.8).

$$\nu_\zeta = C_\mu \zeta k T$$  \hspace{1cm} (B.5)

$$\frac{Dk}{Dt} = \Phi - \varepsilon + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]$$  \hspace{1cm} (B.6)

$$\frac{D\varepsilon}{Dt} = \frac{C_{\varepsilon 1} \Phi - C_{\varepsilon 2} \varepsilon}{\tau} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]$$  \hspace{1cm} (B.7)

$$\frac{D\zeta}{Dt} = f - \zeta \frac{k}{\kappa} \Phi + \frac{\partial}{\partial x_k} \left[ \left( \nu + \frac{\nu_t}{\sigma_\zeta} \right) \frac{\partial \zeta}{\partial x_k} \right]$$  \hspace{1cm} (B.8)

The elliptic equation for the relaxation function, $f$, is given by Eq. (B.9), while the turbulence time and length scales are given by Eq. (B.10) and Eq. (B.11), respectively.

$$L^2 \nabla^2 f - f = \frac{1}{\tau} \left( c_1 + C'' \frac{P_k}{\varepsilon} \right) \left( \zeta - \frac{2}{3} \right)$$  \hspace{1cm} (B.9)

$$\tau = \max \left[ \min \left( \frac{k}{\varepsilon}, \frac{0.6}{\sqrt{6} C_\mu |S| \zeta} \right), C_\tau \left( \frac{\nu}{\varepsilon} \right)^{1/2} \right]$$  \hspace{1cm} (B.10)

$$L = C_L \max \left[ \min \left( \frac{k^{3/2}}{\varepsilon}, \frac{k^{1/2}}{\sqrt{6} C_\mu |S| \zeta} \right), C_\eta \left( \frac{\nu^3}{\varepsilon} \right)^{1/4} \right]$$  \hspace{1cm} (B.11)

The coefficients used are: $C_\mu = 0.22$, $\sigma_\zeta = 1.2$, $c_1 = 0.4$, $C'' = 0.65$, $C_\tau = 6$, $C_L = 0.36$ and $C_\eta = 85$.

B.3 Wall treatment

When simulating wall bounded flows, it is important to have an appropriate treatment for the near-wall flow. The near-wall temperature treatment was described in Chapter 3, so here only the treatment of near-wall velocity in a turbulent flow is described.

B.3.1 Standard velocity wall functions

The most commonly used, "standard" wall treatment is divided into two parts, low- and high Reynolds number formulation. When setting up the simulation, one usually decides upon which treatment to use, having built up the computational mesh accordingly. Figure B.1 shows how the profile of the mean velocity ($u^+$) behaves in the turbulent boundary layer near the wall.
B. Important Governing Equations

Figure B.1: Mean velocity ($u^+$) distribution in the turbulent boundary layer.

\[ u^+ = \frac{u}{u_\tau} \]  
\[ y^+ = \frac{yu_\tau}{\nu} \]  
\[ \mu_w = \frac{y^+}{u_\tau}\mu \]  
\[ u_\tau = \sqrt{\tau_w/\rho} \]

For the viscous sub-layer, when $y^+ < 5$, the velocity is given by

\[ u^+ = y^+ \]

For the log-law layer, when $y^+ > 30$, the velocity is given by

\[ u^+ = \frac{1}{\kappa} \ln(Ey^+) \]

B.3.2 Compound wall functions

Popovac & Hanjalic proposed a revised wall treatment in 2007, named *compound wall treatment*. With this treatment it is not longer necessary to decide on which "standard" treatment to use, high or low Re number treatment. Instead this treatment ensures a gradual change between the viscous sublayer formulations (low-Re) and the wall functions (high-Re). The viscosity $\mu_w$ is evaluated in the same manner as in Eq. (B.14), while the mean velocity ($u^+$) is given by

\[ u^+ = y^+e^{-\Gamma} + \frac{1}{\kappa} \ln(Ey^+)e^{-1/\Gamma} \]

where the exponent, $\Gamma$, is evaluated by

\[ \Gamma = \frac{0.01(\sigma y^+)^4}{\Gamma + 5\sigma^3 y^+} \]