THESIS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY

in

Thermo and Fluid Dynamics

Principles of Heat Transfer in Internal Combustion Engines from a Modeling standpoint

MIRKO BOVO

Department of Applied Mechanics

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden, April 2014

Principles of Heat Transfer in Internal Combustion Engines from a Modeling standpoint

MIRKO BOVO

ISBN 978-91-7385-992-9

THESIS FOR DOCTOR OF PHILOSOPY

ISSN 0346-718X

Department of Applied Mechanics

Chalmers University of Technology

SE-412 96 Gothenburg

Sweden

Telephone +46 (0)31 772 1000

Chalmers Reproservice

Gothenburg, Sweden 2014

Principles of Heat Transfer in Internal Combustion Engines from a Modeling standpoint

MIRKO BOVO

Thermo and Fluid Dynamics
Department of Applied Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2014

Abstract

Heat losses are a major limiting factor for the efficiency of internal combustion engines. Furthermore, heat transfer phenomena cause thermally induced mechanical stresses compromising the reliability of engine components. The ability to predict heat transfer in engines plays an important role in engine development. Today, predictions are increasingly being done with numerical simulations at an ever earlier stage of engine development. These methods must be based on the understanding of the principles of heat transfer.

This work presents the principles of thermo-fluid dynamic behind heat transfer phenomena relevant to internal combustion engines. The emphasis is on the relations between heat transfer and fluid flows. The work provides an overview of the flow-fields characteristics of internal combustion engines. The different approaches to accurate three-dimensional transient modeling of heat transfer and fluid flow are introduced and compared. This information is the backbone to select an appropriate simulation strategy for heat transfer related problems in internal combustion engines.

This thesis presents specific research assessing the ability of numerical simulations to capture heat transfer in complex flows. The single-pulse impinging jet was chosen for its relevance to engine technology and for the challenge it presents to modeling. An ad-hoc experiment was designed for the purpose of validating the modeling approach. The results show how it is possible to use numerical simulations to study heat transfer in complex flow configurations with success.

Publications summary and distribution of work

The present work originates from the increasing need to understand and predict thermal flows in internal combustion engines. Nowadays, in engine development, this task is increasingly carried out with numerical simulations. At an early stage of the Ph.D. work it was decided to focus on the impinging jet-like flame occurring during diesel injection. This phenomenon has interesting heat transfer effects and is challenging to model. The activities carried out can be divided in two parts and are hereby summarized:

PART 1: Numerical predictions of stationary impinging jet heat transfer

The work began by implementing a large number of simulations to evaluate the ability of different steady state CFD models in capturing impinging jet heat transfer. The author realized the models. S. Etemad provided the routines to realize the different mesh topologies tested. L. Davidson contributed with discussions, ideas and reviews. The work was peer reviewed and presented at the "International Symposium on Convective Heat and Mass Transfer in Sustainable Energy", 2009:

1. Mirko Bovo, Sassan Etemad and Lars Davidson "On the Numerical Modeling of Impinging Jet Heat Transfer".

The work was then extended to transient CFD models. L. Davidson contributed supplying the time-space resolved turbulence necessary to properly implement LES. The work was peer reviewed and presented at the conference "Turbulence, Heat and Mass Transfer 7", 2012":

2. Mirko Bovo and Lars Davidson "On the transient modelling of impinging jets heat transfer. A practical approach".

The material published in the conferences was re-elaborated, extended and published in the journal "Numerical Heat Transfer – Part A", 2013:

3. Mirko Bovo and Lars Davidson "On the Numerical Modeling of Impinging Jets Heat Transfer - A Practical Approach", (appendix PAPER 1).

PART 2: Single-pulse impinging jets heat transfer

It was observed that the impinging jet-like flow occurring in a diesel engine differs from the cases available in literature in a number of important characteristics. An experiment was designed to reproduce and measure an impinging jet relevant to diesel injection. The final goal was to reproduce the event with simulations.

The author realized the experiments and ran the respective simulations. The experiment included a number of measuring techniques: PIV, instantaneous thermocouples and IR camera. M. Golubev, E. De Benito, B. Rojo and C. Jimenez Sanchez joined the activities providing essential support when the specific expertise was necessary to run the experimental apparatus. A summary of the work was peer reviewed and presented at the "SAE international conference", 2013:

4. Mirko Bovo and Borja Rojo "Single Pulse Jet Impingement on Inclined Surface, Heat Transfer and Flow Field", (appendix PAPER 3)

The experimental results were collected and published in the European Physics Journal:

5. Mirko Bovo, Borja Rojo and Maxim Golubev "Measurement of a Single Pulse Impingement Jet. A CFD Reference". (appendix PAPER 2)

Finally, one of the experimental setups was investigated with multiple LES runs to perform a quantitative-statistic comparison with the experimental results. L. Davidson made this publication possible contributing with discussions and reviewing. The study is to be submitted for journal publication:

6. Mirko Bovo and Lars Davidson "Using LES to replicate an experimental study of single-pulse impinging jet". (appendix PAPER 4)

The author is completing an industrial PhD sponsored by the Swedish Energy Agency and Volvo Car Corporation. During the entire course of his postgraduate education the author has actively participated in numerous industrial R&D projects strongly related to the subject of this thesis. The knowledge from this experience, combined with the academic work, resulted in the form and content of this thesis.

Acknowledgements

I would like to start from the beginning and thank Sassan Etemad, since without his help I would have probably not even started this venture. Thereafter, much credit goes to all those who believed in me and my project, first of all Prof. Ingemar Denbratt, followed by my managers at Volvo Car Corporation: Peter Norin, Anders Thorell, and Börje Grandin.

During my work there has never been any lack of interesting, constructive discussions, most often concluded with a laud laugh. Mattias Ljungqvist and Joop Somhorst provided much help supervising me from the industrial side. Anne Köster, Jon-Anders Bäckar and Anders Carlsson were sources of much inspiration in the academic environment. All of these wonderful persons were never more than one e-mail away.

Down in the lab, Eugenio De Benito Sienes, Lars Jernqvist, Maxim Golubev, Borja Rojo and Carlos Jimenez Sanchez (We) helped me to turn my crazy ideas into reality. Thanks also to Alf Magnusson and his team for being tolerant enough to have me running around the workshop, even though they did not manage to stop me from smoking in the lab.

In the very limited time in which I was not fully immersed in my work, I had to cope with the dark Swedish winter. This suffering was made a pleasure by Giovanni, to which I will never be thankful enough. For the same reason, I would like to thank all my friends, particularly the ones from my climbing community, for never letting go of me.

I am grateful to Mamma, Papa', Laura e Daniele for giving me so much love from so far away. Me manche'!

The most important at last: I am very grateful to the Scandinavian people of Sweden because, better than most other countries, they created, with their culture, the playground for success for many like me willing to do well. For me, this culture expressed itself in the form of the Swedish Energy Agency and Volvo Car Corporation which financed my graduate education. The same culture permeates Chalmers University of Technology. In my specific case, this reality is embodied by my supervisor Prof. Lars Davidson. To put it simply: Lars, you are the best!

Foreword

"In science one tries to tell people, in such way as to be understood by everyone, some thing nobody ever knew before. But in poetry, it's the exact opposite"

P. A. M. Dirac

This is my thesis as a candidate to achieve the title of *Doctor of Philosophy*. The title distinguishes a person knowledgeable/good at *(Doctor)* about the love/passion *(Philo)* for knowledge/wisdom *(Sophiae)*. Together with the technical content, in this thesis I hope to express my passion for knowledge. Hopefully, the reader will judge my knowledge sufficient and grant me the title of Doctor in this passion of mine.

In this work I try to collect and discuss the principles necessary to successfully simulate heat transfer in internal combustion engines. I decided to treat this subject in general, rather than focusing on simply summarizing the specific work I have already carried out and published. One reason for this is that I spent the last years actively involved in engine development at Volvo Car Corporation. During this period I learned a number of notions that do not connect directly to my specific academic work. Nonetheless, these notions have relevance for the area of my graduate education. I wished to exploit this occasion to collect what I have learned, both for myself and to make it available to others.

I decided to write in the clearest way I can. I wish to make this work accessible to as many as possible. I avoided overly complicated terms and forms. Furthermore, I aimed to collect the ideas concisely, producing a short report. In doing so, I hope to give to the reader more room to elaborate his own thoughts on the notions I discuss.

Symbols and acronyms

C_p	specific heat (J/kgK)
D	diameter / geometrical characteristic length (m)
f	fluid
h	heat transfer coefficient (W/m^{2}K)
k	thermal conductivity (W/mK), or turbulent kinetic energy (m^2/s^2)
$\mathscr{L}$	eddy length scale (m)
l	turbulent length scale (m)
Nu	Nusselt number (-)
Pr	Prandtl number (-)
$\dot{q}$	heat flux (W/m^2)
Re	Reynolds number (-)
T	temperature (K)
t	time (s), or turbulent
T^+	normalized temperature (-)
U	average velocity (m/s)
u	velocity (m/s)
u'	velocity fluctuation (m/s)
U^+	normalized velocity (-)
u_τ	friction velocity (m/s)
v	wall-normal velocity (m/s)
w	wall
x,y	spatial coordinates (m)
y^+	normalized wall distance (-)

μ	molecular viscosity (kg/ms)
δ	boundary layer thickness (m)
ε	turbulent energy dissipation rate (m^2/s^3)
ρ	density (kg/m^3)
τ	shear stress (N/m^2)

CFD	Computational Fluid Dynamics
LES	Large Eddy Simulation
URANS	Unsteady Reynolds Averaged Navier-Stokes

Table of contents

1 Introduction

The purpose of this work is to summarize notions and ideas relevant to the simulation of heat transfer in internal combustion reciprocating engines (ICE, later, simply engines). More specifically, it concerns the heat transferred from the thermodynamic cycle to the solid boundaries of the engine. The main intention is for it to serve as background information necessary in numerically approaching this technological challenge, particularly when using time-resolved three-dimensional simulations. Furthermore, this work is aimed at an audience possessing good knowledge of engine technology.

ICE engines have been an established technology for many years. They have some key advantages that are still unchallenged by today's alternative technologies. The two most relevant characteristic are: the high specific power and the high specific energy achieved with the combination of an engine and a liquid-fuel tank. Figure 1 compares engines to other energy storage technologies. Furthermore, the long history of engine technology adds high reliability and low production costs, making engine technology the first choice for automotive applications so far. For these reasons, engines are not likely to be rapidly phased out, but rather to be used in combination with other propulsion systems.

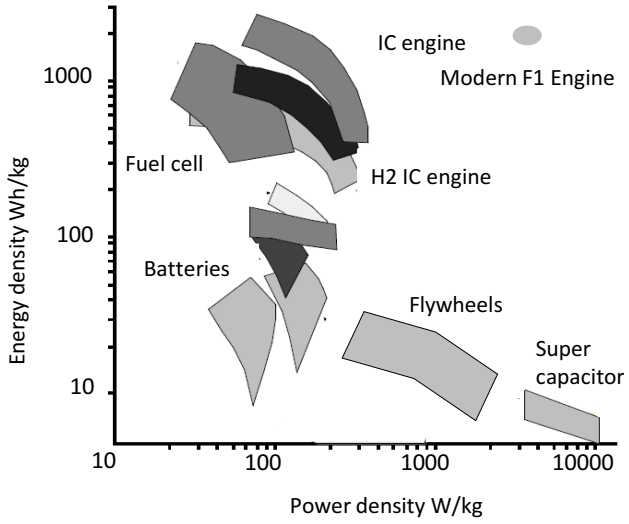

Figure 1: Specific power vs. specific energy, comparison of different technologies. Source data from [1].

Most of the research devoted to engines focuses on the combustion process, and rightly so. The combustion is indeed the driving event for the whole system and deep understanding of this process is the most likely way to achieve key improvements. In engines a combustion chamber is formed in the volume trapped between a closed cylinder and a piston. At each power stroke a charge of fuel and oxidizer ignites. This

initiates an energy flow, from chemical to thermal to mechanical, the latter associated with piston motion.

Engines can be built based on different thermodynamic cycles, the most popular being Otto and Diesel, each with a specific combustion strategy. Regardless of the combustion strategy, the key issue is at a microscopic level. For an efficient reaction, the fuel and the oxidizer need to be in the right conditions (e.g. concentration and temperature) to react and produce exhausts with the lowest possible energy. The largest part of the process takes place in the combustion chamber core and this has been the main focus of engine development for as long as engines have existed. The improvements have been radical, bringing to today's high pressure direct injection systems with controlled charge motion. Consequently, the relative effect of the chamber walls on the charge has increased. This region is characterized by high temperature gradients creating non-optimal conditions for combustion (ignition miss, quenching, slow flame propagation / combustion).

The combustion process has the net effect of providing the heat source to the thermal machine. The process of converting thermal energy into mechanical energy has intrinsic thermodynamic limitations, which might be wrongly interpreted as losses. The actual thermal loss in the combustion chamber is the heat flow through the walls. The combustion process reaches temperatures in excess of 2000K and no conventional material is suitable to manufacture an adiabatic chamber with moving parts. Today, the only practical alternative is to use metals and control the components temperature with an appropriate cooling system. This implies that the temperature of the combustion chamber walls is considerably lower than the core. Consequently, a significant part of the generated heat is carried away through the cooling system with no benefit. The heat transfer between the charge and the walls is dominated by forced convection [2]. Nevertheless, this is no typical internal flow. The motion of a fluid fully confined in a closed cavity with changing volume is almost unique to engine technology.

The interactions between the charge and the chamber walls are many and different in nature. To gain a deeper understanding it is a good exercise to consider in isolation the thermal effects of the different flow-features present in engines. The initial flow momentum is given during the induction stroke by the piston motion. The shape of the intake channels induces certain flow structures (tumble and swirl). With the compression stroke, these structures are deformed or disrupted to create an appropriate flow field at the end of compression. Thereafter the combustion process begins. This does not occur simultaneously in the entire chamber. In this respect the Otto and the Diesel cycles are quite different. In the Otto cycle the air/fuel charge is premixed, the spark-plug locally initiates the reaction that propagates in all directions until it reaches the wall. Differently, in modern Diesel engines, at the end of compression, high pressure fuel is injected in the combustion chamber and self-ignites. The high pressure gives the liquid fuel a high momentum, resulting in a heterogeneous reacting jet-like flow. This eventually impinges on the wall with features much different than any other flow in the engine. For both Diesel and Otto cycles the combustion has the result of rapidly increasing the temperature and, consequently, the pressure inside the combustion chamber. The force generated is then converted into useful work with the piston motion (power stroke). At this stage the surfaces exposed to the charge experience the highest thermal load due to the

combination of high temperature and high flow velocity. The power stroke is followed by the expulsion of the exhausts via the exhaust port. The products of combustion, still at high temperature, are pushed by the piston through the exhaust valve causing yet another flow pattern.

The heat lost by the thermodynamic cycle is the heat load to the mechanical structure. As mentioned earlier, the engine is manufactured using metals. Uneven and transient temperature distribution causes non-uniform thermal expansion of the material and generates mechanical stresses within and between engine components. These components have limits as to the mechanical loads they can withstand. High frequency and low frequency loads act simultaneously on engine components, resulting in a pulsating load which leads to failures by fatigue. Low frequency loads are associated with engine warm-up and cool-down at each usage. High frequency loads are generated by the sudden pressure and temperature increase associated with the combustion at each engine cycle.

In modern Diesel engines, fuel injection results in a high momentum jet-like flame that impinges locally on the piston top. Among the fluid-solid thermal interactions between the charge and the walls, the jet-like flame impingement is identified as the most intense. This flow exists in the engine for a relatively short time, consequently, it is not the major contributor to heat losses. On the other hand, the impingement effects are very localized and capable of inducing intense thermal stresses. Another characteristic is that the thermal effect of impinging jets is particularly difficult to predict with numerical simulations. These two characteristics led the author to choose the study of this flow as a relevant exercise to approach the more general challenge of predicting fluid-structure thermal interaction in engines.

The thermal interaction between the charge and the engine extends both upstream and downstream of the combustion chamber. In the intake and exhaust systems there are components heavily affected by thermal stresses, particularly on the exhaust side. The modeling concepts discussed in this work are extendible to those regions too, as the phenomena are governed by the same physics.

Numerical simulations are rapidly gaining importance in engine development. They provide an increasingly reliable alternative to a number of physical tests, saving costs and time. Numerical simulations can give valuable information only if based on the correct understanding of physics. The laws governing the event of interest need to be known and implemented correctly. The modeling process can be endlessly improved. Nevertheless, the engineering approach limits the complexity according to the purpose of each specific application. Therefore, for a successful engineering approach, it is necessary to have a good knowledge of the available models, their limitations and strengths. This work attempts to provide help in the choice of the most appropriate approach given a specific case.

The content of the work follows this order: the relevant heat transfer and fluid mechanics principles are addressed; the description of the engine cycle is used to isolate the different types of flow structures and their thermal interactions with the solid boundaries; the modeling methods and concepts are described considering their relevance to engine application. These can be seen as the cornerstones necessary for a successful simulation

of engine heat transfer. An additional chapter summarizes the contribution of the author to the field, i.e. a study of the ability of numerical simulations to predict impinging jet heat transfer.

2 Principles of heat transfer

Heat is the transfer of thermal energy. On a microscopic level, thermal energy is associated to the vibrations of atoms and molecules. Hence, it can be seen as a form of kinetic energy. Temperature is indeed a measure of these vibrations. The heat transfer between two bodies at different temperatures is indeed an exchange of kinetic energy at a microscopic level. The high-temperature body passes energy to the low-temperature one, eventually achieving thermal equilibrium. The tendency to thermal equilibrium, or even distribution of kinetic energy, is an expression of the second law of thermodynamics, the driving force of heat transfer.

Matter is discrete, but for practical purposes, in human-scale objects, it is convenient to treat it as a continuum. This way it is possible to think in terms of elements and for these to write macroscopic relations. An *element* is defined as a finite quantity of matter representable with a single value for a given property, for example temperature T. The size of the element is clearly dependent on the specific problem at hand.

This chapter presents the basic mechanisms of heat transfer and it is the first cornerstone to simulate heat transfer phenomena. When working with a complex heat transfer problem, it is important to identify the elementary relations. In other words, it is important to focus on what the element does.

2.1 Heat conduction

Two adjacent elements at different temperatures exchange heat via conduction. This is *always* true regardless of the nature of the elements. The heat flux $\dot{q}$ is described by Fourier's law of heat conduction, in mono-dimensional differential form:

$$\dot{q} = -k\frac{dT}{dx} \tag{1}$$

The thermal conductivity k (W/mK) is a material property. k is in general not a constant and changes with the material conditions, among which temperature itself. Nonetheless, k does not change very rapidly and can be assumed (with caution) to be practically constant in many cases.

2.2 Heat convection

Convection is the mode of heat transfer of fluids within themselves and with their solid boundaries. Fluids are continua for which the relative position between elements is held with relatively weak forces. Both gases and liquids are fluids. For these continua there is a transport of heat associated with the motion of the elements. Heat is indeed *conveyed*. This is far more challenging to predict than the ever-present heat conduction. Indeed, to solve this problem it is necessary to know *how* the elements move. In other words, convection is a property of the flow, not of the fluid.

A first attempt to deal mathematically with heat convection between a flow and its boundary is owed to Newton with his equation of cooling

$$\dot{q} = h\,(T_w - T_f) \tag{2}$$

T_w and T_f are the wall and fluid temperatures. This equation introduces the concept of convective heat transfer coefficient h (W/m^{2}K). h cannot be determined from the nature of the fluid alone, but it is a property of the flow: it is therefore case-dependent and, for most cases, extremely difficult to predict or measure.

The Nusselt number *Nu* is a quantity derived using dimensional analysis. In symbols:

$$Nu = \frac{hD}{k} \tag{3}$$

h is one of the factors in the equation, making *Nu* a property of the flow as well. D (m) is a length characteristic of the geometry at study, for the typical example of a pipe, D is the diameter. As a dimensionless quantity, *Nu* is of great help to transfer findings between similar cases. Nonetheless, the first step in understanding heat convection is the understanding of fluid flows. For most practical cases this means dealing with turbulent flows.

2.2.1 Turbulence

When a force is applied to a fluid element, this moves following Newton's law of motion. Navier and Stokes isolated all the terms contributing to Newton's law for fluid flows. The most relevant properties of fluids in the law of motion are density, ρ (kg/m^3), and molecular viscosity, μ (kg/ms). Density is the resistance of a fluid element against a change in velocity (inertia). Viscosity describes the resistance of the fluid against deformation: it is caused by the friction between fluid elements moving relative to each other.

When studying fluid motion it is helpful to consider the relation between the inertial and the viscous forces. If the inertial forces in the fluid are small compared to the viscous forces, the flow moves in an orderly fashion. These are laminar flows because they behave as if consisting of layers sliding on each other. The heat transfer between sliding layers can be treated as a case of heat conduction. For laminar flows it is possible to fully describe the relative motion of the particles. This can be done analytically for very simple geometries. For more complex geometries it is possible to resolve the flow motion with numerical methods. Solving the flow with the finite volume method allows direct calculation of the flow of mass and heat.

In most relevant technological applications, the flow is in the turbulent regime. When inertial forces affecting a fluid element are comparable to, or larger than, the viscous forces, the flow changes drastically in nature and becomes turbulent. The motion of turbulent flow is very challenging to describe and predict. Although turbulence has been the object of much successful research, it still remains a challenge in modern engineering science.

A useful tool to determine the flow's regime is the Reynolds number *Re*, a dimensionless group derived from flow and fluid properties.

$$Re = \frac{\rho u D}{\mu} \tag{4}$$

This number represents the ratio between inertial and viscous forces. Here D (m) is a representative geometrical length scale, as in the Nu number and u (m/s) is a reference flow velocity. For example, in the case of stationary fully developed flow in a circular straight pipe with no heat transfer, D is the pipe's internal diameter and u is the bulk velocity. For this case, through many experiments, it has been determined that the transition between laminar and turbulent flow occurs between $2300 < Re < 4000$. Notably, this is still only true for practical applications, since under controlled conditions, with very smooth pipes, laminar flow up to $Re = 100000$ has been reached for this case.

Turbulence is therefore a flow characteristic of key importance in understanding and predicting other flow characteristics, such as convective heat transfer. Turbulence can be thought of as formed by small eddies nested in larger eddies. A length scale $\mathcal{L}$ can be associated to each eddy. Each eddy also has a spinning motion and hence a certain kinetic energy. The turbulent kinetic energy k is given per unit mass (m^2/s^2). A useful representation of turbulence is the graph below. Figure 2 represents the turbulent energy spectrum versus the inverse of the length scale. This spectrum is an accepted representation of turbulence and has been verified both experimentally and numerically.

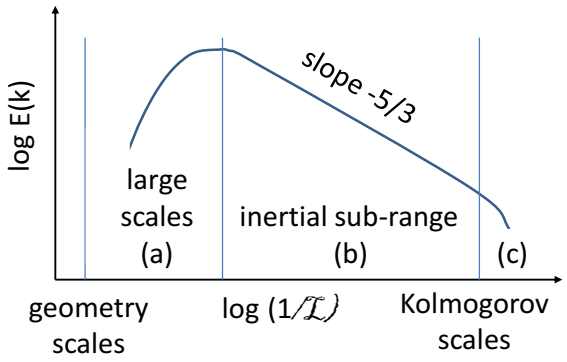

Figure 2: Turbulence energy spectrum.

The turbulent spectrum can be explained taking as an example internal flows. Eddies cannot be larger than the solid boundary containing the flow, hence there is a defined boundary for the plot. Large turbulent structures take energy from the mean flow and transfer most of it to smaller and smaller eddies. This process is known as *"energy cascade"*. In the *inertial sub-range* of the energy transfer is observed to occur with a specific rate corresponding to a slope with coefficient -5/3 as indicated in Figure 2 part (b). In this region the turbulent structures are well represented as isotropic. This means that it is not possible to find persistent identifiable structures. Eventually, the energy is transferred to small eddies for which viscous forces become dominant and, therefore, the eddy-flow becomes dominated by viscosity. This can be appreciated applying the concept of Re number to eddies, based on eddy length scale and velocity. This eddy size is called

Kolmogorov length scale. The viscous forces dissipate the turbulent kinetic energy into thermal energy by friction (part (c) in the figure).

The concepts described above are the foundations of modern turbulence modeling. All the above is strongly related to the convection of heat within the flow itself. To appreciate the heat transfer between fluids and solid parts it is necessary to introduce the concept of boundary layer.

2.2.2 Boundary layer

Practically, any fluid found in engines can be represented with the following correlation in differential form:

$$\tau_w = \mu \frac{du}{dy} \tag{5}$$

This is the definition of Newtonian fluid. In plain words, the wall shear stress τ_w (N/m^2) is proportional to the velocity gradient normal to the wall. The molecular viscosity μ is the constant of proportionality. The wall shear stress is finite and so must be the velocity gradient normal to the wall. At the fluid-solid interface adhesion forces bound the fluid molecules to the wall. Therefore, there is no relative motion between the fluid and the wall at the boundary. This is also known as the *no-slip* condition. Boundary layer theory deals with the velocity profile development in wall-normal direction (i.e. from zero at the wall to flow bulk velocity).

A good case to study boundary layer theory is the velocity profile development on a flat plate immersed in a steady flow (Figure 3). Imagine a thin plate immersed in a moving fluid oriented with the flow streamlines. At the plate's leading edge there is a stagnation point. Here the boundary layer thickness δ (m) is zero. A *Re* number can be associated to the flow using as characteristic length the distance from the leading edge, x (m). A laminar boundary layer grows from the leading edge, "communicating" the presence of the boundary farther into the bulk of the flow. Eventually, due to the increasing thickness of the boundary layer, the viscous interaction between the flow and the boundary is not strong enough to keep the flow laminar. At this point shear forces between adjacent fluid particles in wall-normal direction cause them to roll up, and the flow undergoes transition from laminar to turbulent. This results in an exchange of mass in wall-normal direction, which is a core principle of convective heat transfer for bounded turbulent flows. Moving further along the plate, the mean boundary layer thickness grows at a rate proportional to $\sim x^{4/5}$. Here the flow characteristics remain self-similar with increasing distance from the leading edge. This defines a fully developed turbulent boundary layer. In the region closest to the boundary, the flow is dominated by viscous forces. Consistently with the no-slip condition, velocity develops in a laminar fashion for a certain thickness. This region is called the *viscous sub-layer*. Farther into the boundary layer, the flow presents velocity fluctuations characteristic of turbulent flow. The area where the two zones merge is called the *buffer layer*.

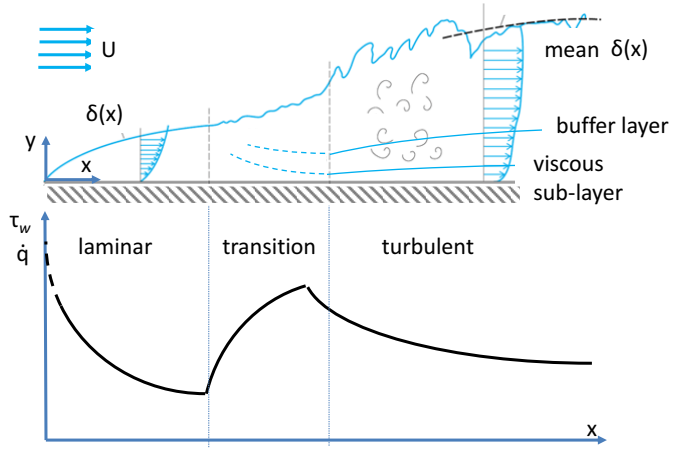

Figure 3: Boundary layer development over a flat plate.

The no-slip condition implies that there is no relative motion between the fluid and the solid at the interface. Consequently the fluid molecules in contact with the wall have the time to reach thermal equilibrium with the solid outer surface. Consequently, both fluid and solid must have the same temperature at the interface. Furthermore, considering the similarity between the equations (5) and (1), shear stress and heat flux have similar behaviors. This is qualitatively illustrated in Figure 3 (bottom) for the flat plate case. At the leading edge the boundary layer thickness is very thin and both the shear force and the heat flux are very high. Moving downstream, the boundary layer grows and consequently the flow bulk is farther from the wall, gradually lowering the gradients driving τ_w and $\dot{q}$. Eventually, the boundary layer transitions from laminar to turbulent. Elements within the boundary layer begin to be exchanged in wall-normal direction, conveying momentum and thermal energy. The result is a sudden increase in both shear force and heat flux. From there on, the boundary layer slowly grows and the interactions between flow and wall weaken once again.

From all the above, it appears clear that if we could know what the boundary layer looks like at any given position, we could apply the heat conduction equation on the viscous sub-layer and solve for heat transfer coefficient exactly, combining Fourier's and Newton's equations (1) and (3).

$$h = \frac{-k \left.\frac{dT}{dy}\right|_{y=0}}{(T_w - T_f)} \tag{6}$$

Further insight on the boundary layer can be gained studying the velocity profile in the fully developed region. The velocity profile and the wall distance can be normalized using friction velocity u_τ. The friction velocity is not an actual velocity, but a quantity with the units of velocity (m/s). The expressions for the normalized velocity U^+ and the normalized wall distance y^+ are:

$$U^+ = \frac{U}{u_\tau}, \quad y^+ = \frac{\rho u_\tau y}{\mu}, \quad u_\tau = \sqrt{\frac{\tau_w}{\rho}} \tag{7}$$

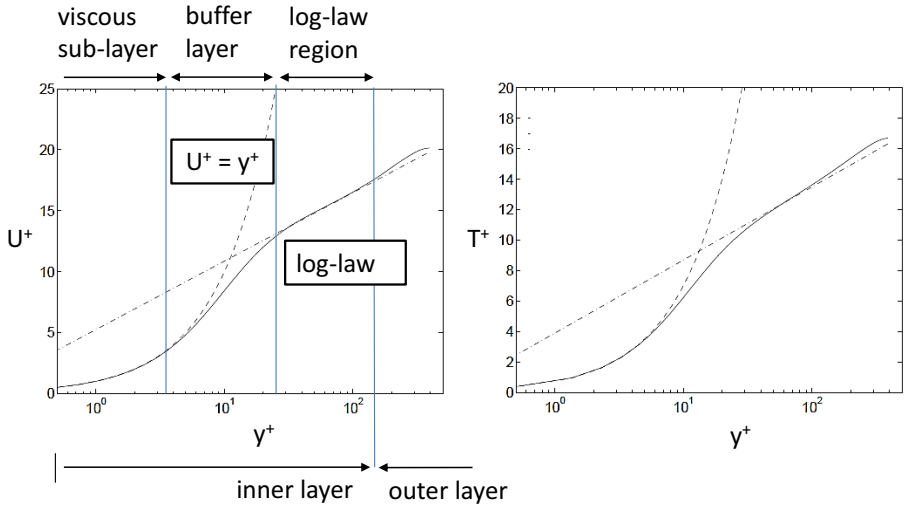

Figure 4: Law of the wall for velocity and thermal boundary layer. Source data from [3].

The information plotted in Figure 4 represents the knowledge we currently have of the near-wall velocity profile for fully developed boundary layers. The concept of *Re* number can be deployed to describe the local features of the flow, this time using as characteristic length the wall-normal distance *y*. Indeed, it is easy to recognize that equation (7), the definition of y^+, has the same expression as the definition of the *Re* number, equation (4). For fully developed boundary layers it is possible to identify representative values of y^+ for which the velocity profile has laminar or turbulent characteristics. This is shown in Figure 4 indicating the laminar, transition and turbulent region, respectively called viscous sub-layer, buffer layer and log-law region. The plot in Figure 4 includes the "law of the wall" which assumes U^+ to be proportional to $\log(y^+)$ in the turbulent boundary region, hence the term *log-law*.

The above approach can be applied to the thermal boundary layer as well (Figure 4 right). In this case the expression for the normalized temperature T^+ is:

$$T^+ = \frac{(T_w - T)\rho C_p u_\tau}{\dot{q}} \tag{8}$$

with C_p (J/kgK) being the fluid specific heat, T_w the wall temperature and $\dot{q}$ the wall heat flux. Figure 4 shows a clear similarity between the velocity and the thermal boundary layer. This similarity is often used to model the two interconnected phenomena.

The flat-plate case is a largely simplified example which helps in understanding boundary layer theory, but it is rarely found in practice. In actual applications a number of flows are

found, each with its own particular boundary layer. Examples are presented in Figure 5 where it is possible to appreciate different phenomena, such as stagnation points, separation points and multiple wall interactions. From the considerations above, the complexity of such flows in the near wall region should be apparent. Consequently, the prediction of the heat transfer associated with these flows is complex.

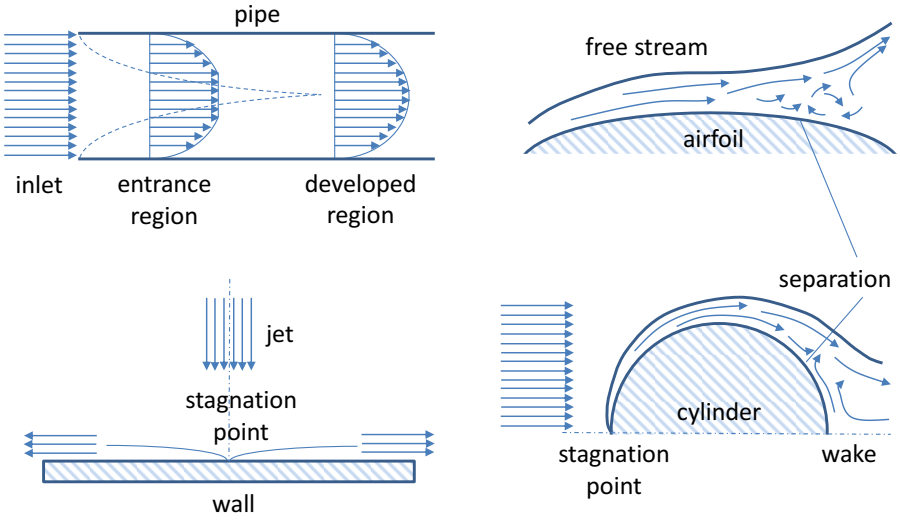

Figure 5: Example of boundary layer in complex flows.

One further complication arises from wall roughness. A rough surface can be imagined formed of peaks and valleys. If these formations have a size similar to, or larger than, the size of the viscous sub-layer, they will interact with the boundary layer development with consequences on both the velocity and the temperature profiles.

2.2.3 Heat transfer correlations

Using dimensional analysis it is possible to derive the Prandtl number *Pr*. This dimensionless quantity is useful in the study of convective heat transfer. It is defined as:

$$Pr = \frac{C_p \mu}{k} \tag{9}$$

The *Pr* number is a property of the fluid expressing the ratio between the viscous and the thermal diffusion coefficient. Physically, strictly for laminar flows, it represents the ratio between velocity and temperature boundary layer thickness.

For stationary fully developed flows it is possible, analytically or experimentally, to derive correlations for the convective heat transfer. These can be of use in the solution of a few simple cases, or as a rough first engineering approximation. These relations usually have the following form:

$$Nu = a\, Re^b\, Pr^c \tag{10}$$

with a, b, c constants usually determined experimentally. Unfortunately, for most engineering solutions and, certainly for engines, the fluid flows are quite complex. A number of correlations for engine heat transfer exist: these are useful, but they are very limited in accuracy and applicability.

2.3 Heat radiation

Heat is transferred between bodies at different temperature also via electromagnetic waves, a phenomenon called *heat radiation*. This is a surface phenomenon for solids but it is also present in fluids containing non-transparent molecules, such as carbon oxides and soot. This work does not include this form of heat transfer. Nevertheless, it is important to mention its existence. Radiation is an important, yet secondary, contributor to the overall engine heat transfer. The relevant characteristic of radiation for this work is that it occurs simultaneously and independently in relation to the other modes (conduction and convection). As the other modes of heat transfer, heat radiation affects the temperature field, but its effects can be accounted for separately.

2.4 Transient heat transfer

In general, the temperature of an element is not constant in time. For example, solid parts can be exposed to varying thermal loads. Even at stationary thermal conditions, fluid elements in the flow move between areas at different temperatures. The temperature change in time of an element is determined by its energy balance. This depends on two different properties. On one side, how the element conducts thermal energy, thermal conductivity k, and on the other, the amount of energy necessary to alter the element's temperature, ρC_p. These can be grouped to define thermal diffusivity α (m^2/s):

$$\alpha = \frac{k}{\rho C_p} \tag{11}$$

3 Engine flow from a heat transfer point of view

Knowledge of the flow nature is essential to its successful simulation. This chapter focuses on heat transfer between the charge and the engine walls containing it. Charge motion and charge temperature evolution have a large effect on heat transfer. On the other hand, wall temperature has only marginal effects on the flow pattern and on the chemistry of the main flow. In other words, the thermal interaction between the charge and the engine walls can be considered to be *one way coupled*.

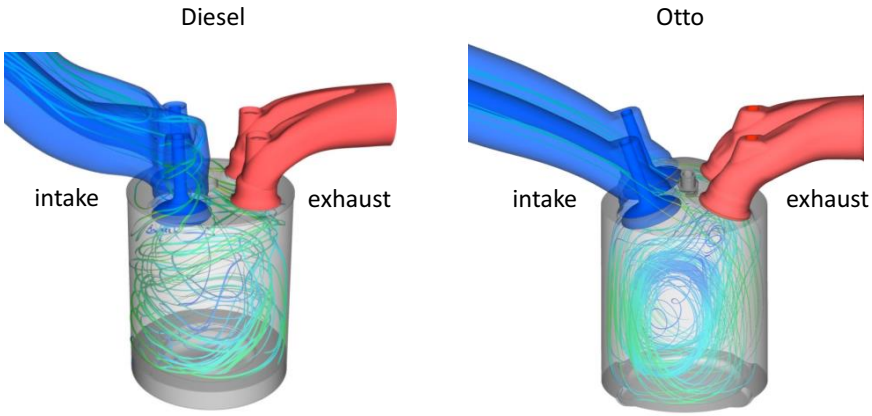

Figure 6: Example of in-cylinder charge motion at end of intake stroke. Courtesy of Mattias Ljungqvist, Volvo Car Corporation.

Figure 6 gives a graphical overview of the flow inside an engine. Looking at the picture, keeping in mind the physics discussed in the previous chapter, the challenge posed by performing simulations of engine flow is apparent. To put it in words, it is a multidimensional, time-dependent internal flow induced by a geometry change. It is not wise to simulate such a flow at once. To assist in choosing the appropriate modeling strategy, it is useful to identify the most relevant flow features: this is done in the following sections.

3.1 Combustion chamber flow

The most important event in the engine is the combustion process in the cylinder. The chemical process causes the charge to reach its highest temperature, thus making the flow inside the chamber of primary importance for the study of heat transfer. The combustion chamber is a closed environment with changing geometry due to piston movement. Charge motion is initiated with the intake stroke, which transfers momentum to the flow. Thereafter, with closed valves, piston upward movement reduces chamber volume and affects charge motion. Heat transfer between the charge and the walls is enhanced by their relative motion. At the same time, charge motion is of primary importance to the combustion strategy. Briefly, a good combustion relies on bringing in contact fuel and oxidizer at the right time and at the right rate. Therefore, different combustion strategies favor different charge motions. There are two major types of in-cylinder motion: tumble

and swirl. Although these coexist simultaneously, the engine design can favor one over the other. This can be done guiding the flow during the intake stroke to form coherent structures. An example of tumble and swirl can be seen in Figure 6 while Figure 7 schematically shows the principles of the different charge motions.

In general, tumble is preferred for homogeneous spark ignition combustion systems, running most often with gasoline and based on rapid flame propagation. The goal is to control ignition-timing with a spark and thereafter proceed to full combustion as quickly as possible. Tumble contributes first to the homogenization of the charge involving the entire combustion chamber. Thereafter the geometrical size of the tumble motion is no longer compatible with the reducing height of the chamber and the flow breaks down into smaller and smaller vortices. The result is a highly turbulent flow that increases the propagation speed of the flame front.

Swirl is favored by compression ignition combustion systems. These run most often with diesel fuel and are based on the diffusion controlled combustion. A swirling motion is induced during the intake stroke. Thereafter the chamber volume decreases, but the charge motion is well oriented to fit the chamber's shape and the large structure can survive intact. Towards the end of compression the peripheral crown of the piston comes very close to the cylinder's head and generates a squish of flow towards the center of the cylinder. The piston's top has a characteristic bowl shape that is designed to accommodate the swirling flow. The charge's angular momentum must be conserved, hence reducing the radius of the swirling charge causes a drastic increase in angular velocity. Simultaneously, the flow is curled up by the inward motion. From the center of the cylinder head, high pressure liquid fuel is injected, aimed at the bowl lip. Suddenly exposed to a high temperature environment, the fuel rapidly evaporates and self-ignites. The fuel spray, with its high momentum, disrupts the swirl structure creating a highly turbulent environment that favors the combustion process.

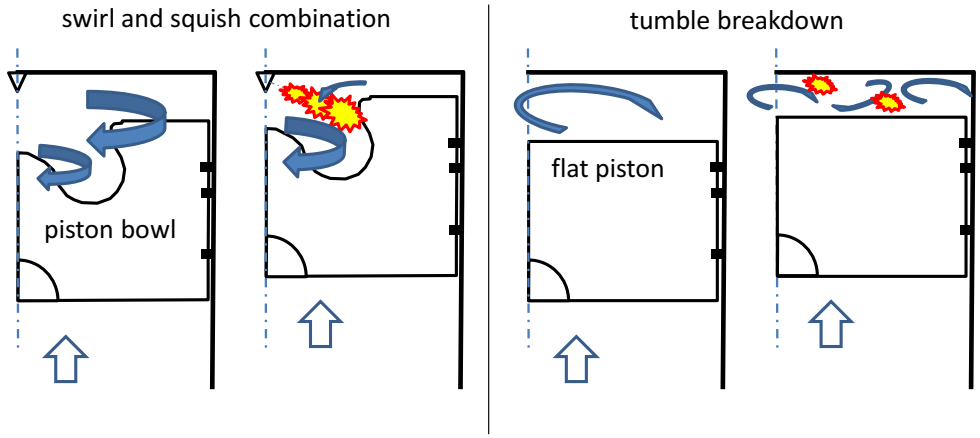

Figure 7: Tumble, squish and swirl charge motion.

From the description above it is evident that the flow in the combustion chamber is quite unique and so is the heat transfer associated with it. In general, the cylinder walls

experience an ever-changing flow which does not allow for the growth of a well-defined boundary layer. Moreover, there are no clear locations suggesting the formation of stagnation points or flow separation. Therefore, at the wall, there is a boundary layer with a finite thickness which constantly evolves with time and location during the cycle. This is confirmed by experimental observation [4].

During compression the charge increases its temperature uniformly. This is not true during the combustion process. Otto and Diesel have different combustion strategies which lead to different effects. For Otto engines, the combustion propagates rapidly from the spark plug through the whole charge. However, for a short period of time, different cylinder surfaces are exposed to fluids with very different temperatures, namely the burned and unburned charge. This significantly affects the local heat flux. Later in the power stroke, the gases mix and the fluid's properties are uniform once again. The general discussion above can be applied to the Diesel cycle with one important exception: fuel injection. This deserves to be discussed in a separate section.

3.2 Diesel injection (single-pulse impinging jet)

In Diesel engines a single-pulse jet-like flow is generated by the injection process. High pressure liquid fuel is injected in the center of the combustion chamber through small holes and it is aimed at the surface furthest away (Figure 8). High injection pressure induces high velocity in the liquid jet, which plays a key role in spray formation. The shear forces between the liquid and the surrounding charge cause the jet core to break up, enhancing evaporation and mixing between the fuel and the oxidizer. In many cases, particularly at high load, this jet-like flame impinges on the chamber wall causing unwanted effects.

Considering the combustion process, the jet locally reaches self-ignition conditions and the combustion process begins. The diesel flame is dominated by diffusion with the flame front enveloping the fuel-rich core and exceeding temperatures of 2000K. A description of diesel combustion is given by [5]. Although the air in the combustion chamber has a relatively high density and momentum (swirl), fuel-jet momentum largely overrides the environmental effects and evolves into a jet-like flame which eventually impinges on the piston bowl/top. Figure 8 (left) shows pictures of diesel injection taken with different techniques, and they clearly support this description. The flame impingement has important consequences on the combustion. The flame front approaching the wall runs out of fresh air, thus locally reducing the oxygen available for the reaction. Consequently, the combustion process slows down and the jet is left with large amounts of intermediate combustion products, such as soot precursors. Furthermore, the contact with a cold surface gives the soot precursors the conditions to aggregate and form large particles [5 and 6].

Impinging jet flow has noteworthy heat transfer effects. In the impingement region, the flow splits to form a wall-jet. At the split point the wall parallel velocity is zero. This point is a stagnation point. From here, the boundary layer develops alongside with the formation of the wall-jet. At the stagnation point, boundary layer thickness is "zero" (extremely thin), bringing locally the flow bulk close to the surface. The consequence is a

high localized heat transfer. In the case of diesel injection, the temperature gradient between the gas and the piston surface is extremely high and, consequently, so is the local heat flux. In diesel combustion the matter is further complicated by the fact that the jet is delivered as a pulse. In this case the heat transfer relation is also a function of time.

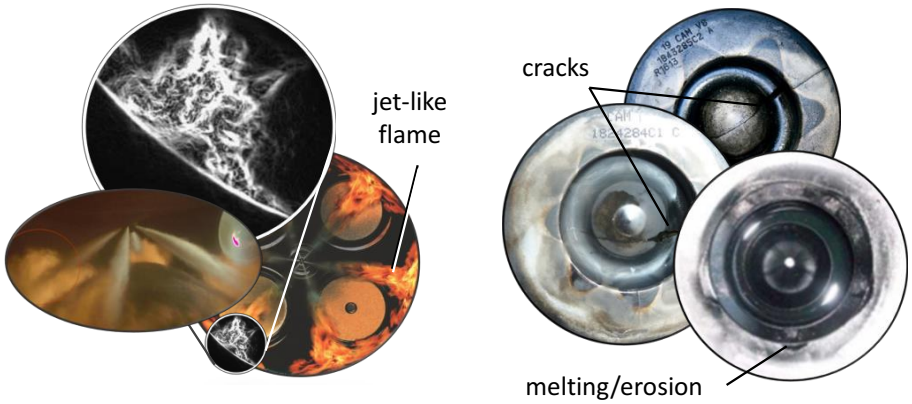

Figure 8: (Left) Impinging jet flow resulting from diesel injection in engines. Image compilation sources: [7] and internet search. (Right) Diesel piston tops damaged by local thermal stresses due to jet impingement (cracks and melting/erosion).

Although the impinging jet is the event causing the highest heat flux, it is localized and short-lived compared to the rest of the fluid-solid interactions. Therefore it is not the largest contributor to engine heat losses. The impinging jet is instead cause of large localized temperature gradients on the piston top. The high frequency of such thermal interaction on the piston surface is the cause of thermally induced mechanical fatigue. This is known to be a cause of engine failure. Figure 8 (right) shows pictures of Diesel piston tops. The effect of jet impingement is clearly visible with local melting/erosion and cracks.

3.3 Intake and exhaust (ducts, manifolds, ports and valves)

The fresh charge reaches the combustion chamber via the intake duct. The flow splits in the intake manifold where it is directed to the different cylinders. Approaching the intake valves, the flow is pulsating with increasing intensity being affected by the cylinders phases. The charge is inducted into each cylinder during the intake stroke, which lasts roughly ¼ of the cycle. The fluid forms therefore a transient internal flow in the duct.

The intake valves are part of the intake system. The upper surface of the valve head, together with the port wall, forms a convergent geometry and experiences an accelerating internal flow. The lower part of the valve head is effectively part of the combustion chamber. The stem of the valve is an inclined cylinder inside the intake port, which experiences a pulsating external flow. Flows around cylinders present interesting heat transfer characteristics. A boundary layer develops from the cylinder front, grows along the cylinder surface and eventually separates creating a wake behind the cylinder. Consequently the heat transfer coefficient varies along the cylinder's surface.

The charge entering the cylinder is in general at a lower temperature than the engine walls. Consequently, the fresh charge provides a small cooling effect to the intake ports and components. This phenomenon does not contribute directly to engine heat losses and might be disregarded studying the overall engine energy balance. On the other hand, heating of the fresh charge lowers its density. The consequence is a reduced volumetric efficiency. Furthermore, knowledge of the heat flux on the intake ports is necessary to solve the entire temperature distribution of the cylinder head, a component which is difficult to design and prone to failure.

In many senses, the flow in the exhaust manifold can be compared to the one in the inlet manifold with the important difference that the charge is at a much higher temperature and pressure. Exhaust and intake valves have practically the same shape, nevertheless they experience rather different flows. For exhaust valves, the gases are pushed out of the combustion chamber against the head of the valve, thereby establishing a stagnation region. The charge then goes around the valve head and creates a highly turbulent wake engulfing the valve stem. Both these flows, impinging and wake, have high levels of heat transfer. This, together with the high temperature gases, makes the exhaust valve a component that is extremely affected by thermal stresses. The problem is worsened by the fact that the cooling of the exhaust valve head is limited. Indeed, the valve head is primarily cooled during the temporary contact with the valve seat during valve close time.

The exhaust valve bridge is the area of the combustion chamber between the two exhaust valves, it is however relevant to mention it in this discussion. During combustion this area experiences the same conditions as the rest of the combustion chamber walls. Once the exhaust stroke begins, this isolated bit of material works as flow-splitter between the two sides of the exhaust port: consequently it is the location of a stagnation zone. Furthermore, due to its location, the exhaust valve bridge is difficult to access with large cooling channels. Consequently, this component is critically stressed by thermal loads and it is a known location for mechanical failure.

3.4 A note about soot

Under running conditions the engine might develop a soot layer on the surfaces in contact with the combustion products (combustion chamber and exhaust system). This layer is in general a good thermal insulator which has the positive effect of limiting heat transfer. Notably, the soot layer can be oxidized and disappear. In Diesel pistons, it could be the case that the soot is locally removed in the jet impingement area. Consequently, there is a further increase in temperature gradient for the solid close to the surface. Increased temperature gradients are associated with increased thermal stresses.

3.5 Engines fluid-solid thermal interaction, graphical overview

The phenomena described above can be summarized in graphical form. Each solid surface in contact with the charge experiences a cyclic combination of flows. This combination is not only dependent on the location but also on the engine load. Figure 9 presents two graphs giving an overview of the temperature evolution of the combustion

chamber near-wall region in time (right) and space (left). The figure is a virtual overlap of two different locations, one is arbitrary and the other is the jet impingement area. Both experience similar flows except during the injection process. The figure plots the qualitative temperature variation of the charge, the thickness of the thermal boundary layer, the temperature of the near wall surface and the temperature a bit within the solid. Charge temperature and boundary layer thickness change with time. The solid temperature closest to the surface will also vary in time, creating a smooth transition between the temperatures at the fluid-solid interface and within the solid. The figure on the right presents the time-resolved temperature variation for the charge bulk and at the fluid-solid interface. The interface temperature follows charge temperature with a certain delay due to its thermal inertia. The cyclic temperature variation of the impingement region is similar to the arbitrary surface, but with a larger temperature variation.

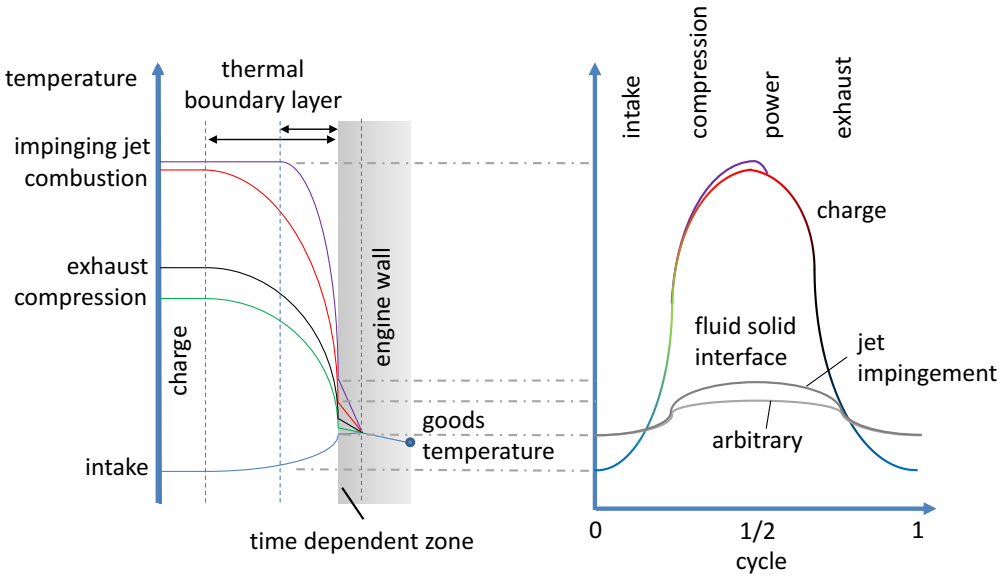

Figure 9: Generic temperature profile in engine surface as function of cycle for two locations (arbitrary and jet impingement).

4 Modeling heat transfer in engines

The purpose of a simulation is to predict the behavior of a phenomenon of interest. Clearly, the first step for successful simulation is to define the problem, this was discussed in the previous chapter. In engineering, a good approach should have two characteristics: it should represent with adequate accuracy the problem at hand, and, it should be simple and fast. For this reason there are virtually as many models as there are applications. Considering engine heat transfer, the simplest models represent the entire engine as a single entity (global models). The models can be refined in space (multi-zone models) and time (transient models). An introduction to these models can be found in [4]. Another degree of refinement can be applied to the model's principles, for example, with how well the model represents the materials properties or the physics of the phenomena.

This work focuses on high resolution three-dimensional transient simulations based on the local thermal interaction between turbulent flows and their solid boundaries. Even this approach can be carried out with different degrees of refinement. This description begins from the finest and most accurate model, and step by step it introduces simpler, faster approaches. The purpose of high resolution three-dimensional transient engine simulations is to predict in detail the charge motion and the related heat transfer. To achieve a valuable representation of such complex phenomena it is necessary to base the complete simulation on a number of models representing different parts of the physics involved. The relevant models describe aspects such as turbulence, boundary layer and geometrical representation. These are individually treated in this chapter. Each approach has its strengths and weaknesses. Good knowledge of modeling principles is one of the cornerstones needed to guide the choice of an appropriate simulation approach for a specific problem at hand.

4.1 CFD principles

The basic idea of CFD is to divide a large complex flow in smaller simpler flows, and to describe them with manageable mathematical expressions. The solution of this system of equations is then a description of the large complex flow. In practice the finite volume method is applied to the equations governing the flow. The equations become then discrete rather that continuous and are solved using numerical discretization methods. The geometrical discretization of the entire flow is called mesh or grid and it is composed of cells or elements, which are the control volumes.

The equations necessary to describe the flow are: continuity, momentum and energy:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j\right) = 0 \tag{12}$$

$$\frac{\partial}{\partial t}\left(\rho u_i\right) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j + p\delta_{ij} - \tau_{ij}\right) = 0 \tag{13}$$

$$\frac{\partial}{\partial t}(\rho e) + \frac{\partial}{\partial x_j}\left(\rho u_i e + u_j p + \dot{q}_j - u_i \tau_{ij}\right) = 0 \tag{14}$$

ρ, u and $\dot{q}_j$are density, velocity and heat flux already introduced in chapter 2. p (Pa) is the pressure, e (J) is the internal energy and δ_{ij} (-) is the Kronecker operator. τ_{ij} (Pa) is the viscous stresses tensor and its expression is:

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij} \tag{15}$$

The equations above describe the instantaneous flow field and different models use different approaches to account for the equations terms. Numerical modeling is based on the discretization of the equations using numerical methods. The variables in the equations are discretized as follows, using as an example velocity:

$$u = U + u' \tag{16}$$

In words, the actual velocity u is the sum of time average U and instantaneous fluctuation u'.

CFD stands for Computational Fluid Dynamics but in the context of this work it is important to explicitly state that the approach can be directly extended to solids. For stationary solids it is not necessary to solve the momentum and continuity equations. Only the energy equation for conduction needs to be solved.

4.2 Modeling turbulence

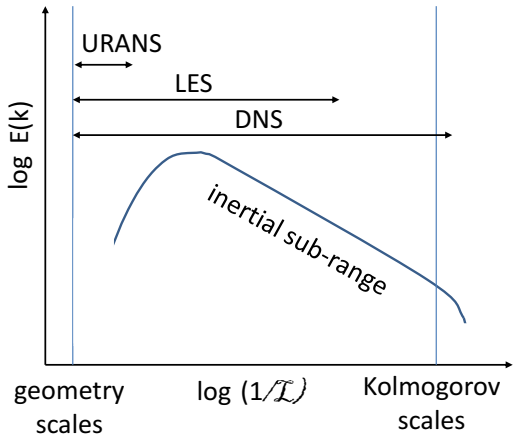

Figure 10: Turbulence models in relation to turbulence energy spectrum.

Figure 10 presents the spectrum of turbulent kinetic energy as function of different turbulent eddy size. This distribution is a general characteristic of turbulent flows and is

discussed in section 2.2.1. The areas where different turbulence models work, are highlighted in the same figure.

At a small-scale level, the physics governing turbulence is well understood and it is possible to simulate turbulent flows with very high resolution. The challenge arises from the strong interaction between very different scales in space and time, a difference that grows for increasing Reynolds numbers. In other words, it is not possible to accurately predict the behavior of a large scale flow without knowing the behavior of the small scales flows present within it. There are different approaches to the simulation of turbulent flows, they differ on how turbulence is modeled.

4.2.1 DNS (Direct Numerical Simulation)

DNS is arguably a model of turbulence. This type of simulations discretizes the computational domain with cells small enough to resolve the Kolmogorov scales (see Figure 10). These elements are in the viscosity dominated regime and they can be resolved exactly. DNS is used in research to study test cases with relatively low Reynolds number but they are yet far from being applicable to engineering problems due to their large computational cost.

4.2.2 LES (Large Eddy Simulation)

LES is an approach to simulate turbulent flow which arises from observation of turbulence energy spectrum in Figure 10. In the inertial sub-range of the energy cascade, the eddy structures are statistically self-similar. The effect of this eddy breakdown is a net transport of turbulent kinetic energy to smaller and smaller eddies. This process can be represented with a relatively simple model known as SGS (Sub-Grid-Scale) model. In a LES simulation the computational domain is divided in cells with size within the inertial sub-range. Within each computational cell turbulence is modeled with a SGS model. The large eddies extracting energy from the mean flow are instead resolved in space and time.

The equations implemented to realize the LES model are available in appendix PAPER 1. The paper is a work by the author, comparing the ability of different turbulence models to predict impinging jet heat transfer. At the core of a good LES simulation there is a proper spatial discretization of the computational domain. It is not always easy to determine the correct cell size to properly capture the large eddies, particularly in flow fields with local characteristics such as recirculation zones, stagnation points or shear layers.

Most SGS models are relatively simple, and in general the more turbulence is resolved, the better the assumptions for the SGS model are satisfied. The Smagorinsky SGS model was the first to be proposed and is the best known thanks to its simplicity. This model is best suited for unbounded flow, since it does not account for the presence of the wall. The turbulent structures close to a solid boundary have different characteristics compared to the ones in the main stream. More recent models are better suited to study the fluid-solid interaction. For example the WALE (Wall Adaptive Local Eddy-viscosity) model accounts for the effect of the wall on turbulence. Models of this type should be preferred when studying heat transfer.

Today LES is used in academia and increasingly in industry. LES has shown many promising results, the limitations are for the most part still related to computational costs. To work properly, LES meshes need to be fine enough to resolve the large eddies, resulting in a large number of cells, for typical engineering applications. Furthermore, engineering most often requires average data. Hence, for stationary flows, the LES solution needs to be run long enough to collect statistically relevant information. For transient cases, such as engine flow, a number of simulations needs to be run to get statistical information about cycle-to-cycle variations.

4.2.3 URANS (Unsteady Reynolds Average Navier Stokes)

The strong advantage of URANS models is that they are far less computationally expensive compared to DNS and LES. This advantage comes at the expenses of accuracy. In today's engine development, URANS simulations largely dominate.

Most URANS are based on the Bousinnesq approximation. This assumes that the effects of modeled turbulence can be accounted for with a scalar isotropic property called turbulent viscosity μ_t (kg/ms). Turbulent viscosity is calculated locally in the computational domain and is related to the local modeled turbulent length and velocity scales. The terms accounting for turbulence production and dissipation are related to the local gradients of the mean flow. These terms appear in the transport equation for the turbulent quantities.

Many models are developed from the assumption above. The "two equations model" family accounts for the effect of turbulence with two transport equations. The most popular of these models for industrial application is the k-ε model. In this model μ_t is derived from the turbulent kinetic energy k and the turbulent dissipation rate ε (m^2/s^3). Transport equations for these two quantities (k and ε) are solved along with the momentum and the energy equations. A description of these models is available in PAPER 1.

A strong limitation of the standard two equations models is that they have no direct way to account for the presence of walls. Therefore, they perform best in the fully turbulent regions and where there are no strong wall-normal interactions. These models can be formulated so as to resolve the boundary layer but in most cases the near-wall region is accounted for with a wall-function, which will be the topic of the next section.

Close to the wall the assumption of isotropic turbulence is not satisfied, the solid boundary has a strong influence on wall-normal velocity fluctuations. The limitations posed on the wall-normal fluctuations in turn affect also the wall-parallel fluctuations. The energy transfer between the fluctuations in different directions occurs via the *pressure strain*. The V2F model reported in appendix PAPER 1 is similar to the k-ε model but it solves for two extra equations, one for the wall-normal Reynolds stress v'^2 (m^2/s^2), and one for the redistribution of v'^2. The V2F model automatically detects the presence of a wall and accounts for its effect on the turbulence. This model is presented and investigated in PAPER 1. It is found to be noticeably better than the k-ε model for impinging jet flows. The V2F model is implemented as a low-Re model for the near wall treatment: this too will be the topic of the next section.

4.3 Modeling velocity and thermal boundary layer

The near wall region poses a further challenge for modeling. The presence of the wall affects flow physics as discussed in section 2.2.2. The assumptions of isotropy and fully developed turbulence are not valid near the wall: for this reason a specific approach is necessary in this region. There are different ways to treat the near-wall region and they can be combined with the different turbulence models.

4.3.1 Low-*Re* wall treatment

If the near wall phenomena are important in the issue at hand, as it is in convective heat transfer, it is better to use a low-*Re* model. The expression "low-*Re*" refers to the turbulent Reynolds number defined as:

$$Re_t = \frac{\rho u' l}{\mu} = \frac{\mu_t}{\mu} \tag{17}$$

where l (m) is the turbulent length scale. The approach is independent of the turbulence model and refers to any model resolving the velocity profile through the boundary layer all the way to the wall including the thin viscous sub-layer. In this region the flow is dominated by viscosity and the velocity profile can be determined exactly. This is true for both the velocity and the thermal profiles. In practice, to resolve the boundary layer it is necessary to have near-wall cells satisfying the criterion $y^+ < 1$ (see Figure 11). This implies using thin cells in the near-wall region. A good mesh must also have a smooth growth in cell size from the wall into the main flow. Consequently, the complete mesh results in a large number of cells. Notably, the larger the *Re* number, the thinner the boundary layer, and the larger the number of cells, for the same geometry.

For LES simulations, low-*Re* approach is necessary by definition to resolve the anisotropic turbulent structures near the wall. To do this, the first cell thickness must satisfy $y^+ < 1$ and the cell growth rate in wall normal direction should be < 1.2. Figure 11 shows a mesh created with these guidelines, alongside the fully developed boundary layer profile for comparison. Furthermore, the cells should have $y^+ < 30$ in the cross flow direction and $y^+ < 100$ in the stream wise direction. With these criteria, the near wall eddies are captured with high enough accuracy for most cases.

With URANS there is no need to resolve the large eddies. Nevertheless, the low-*Re* approach can be used to resolve the boundary layer. Even for these models the first cell must satisfy the criterion: $y^+ < 1$. On the other hand, the cell size in the wall-parallel directions and the mesh growth rate can be considerably larger than with LES, resulting in a significant reduction on the total number of cells.

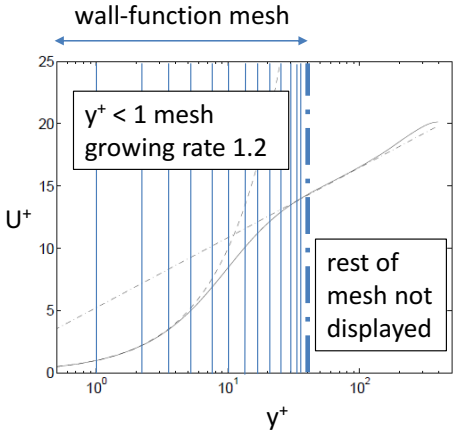

Figure 11: Near-wall mesh relation to boundary layer velocity profile.

4.3.2 High-*Re* wall treatment (wall-function)

A commonly used approach in CFD is the use of wall-functions to reduce the computational effort. Models using this approach are also known as high-*Re* model. Wall-functions are a sensible approach when the main focus of the simulation is the main flow with a well-developed boundary layer. If the boundary layer is developed, the velocity profile is well represented by the law of the wall discussed in section 2.2.2. If the velocity profile is known, there is no need to resolve it, and the whole region's behavior can be represented with a single cell (see Figure 11). If the boundary layer is not developed, as in separation and stagnation zones, high-*Re* models do not perform well. This is sometimes tolerated given the large advantage these models give in computational speed. High-*Re* models should be used with extreme care when implemented in a simulation to predict heat transfer related problems.

The connection between the momentum equation and the energy equation in high-*Re* models can be done using Reynolds analogy. This analogy presupposes that the turbulent momentum diffusivity μ_t/ρ and the turbulent thermal diffusivity α_t are proportional. The definitions of μ_t and α_t are:

$$\mu_t = \rho \frac{\overline{u'v'}}{dU/dy} \quad , \quad \alpha_t = \frac{\overline{v'T'}}{dT/dy} \tag{18}$$

with *u'* the stream-wise velocity fluctuation, *v'* the wall-normal velocity fluctuation and *T'* the temperature fluctuation. The turbulent Prandlt number, Pr_t is defined as:

$$Pr_t = \frac{\rho \mu_t}{\alpha_t} \tag{19}$$

The velocity and the temperature boundary layer profiles were shown to have strong similarities in 2.2.2 and the turbulent Prandlt number is used to relate the velocity and the temperature boundary layer thickness. A deeper discussion about the wall boundary

conditions for turbulence models, including the arguments for the Reynolds analogy, can be found in [3].

The presence of the wall has important effects, particularly limiting wall-normal turbulent fluctuations. One effect is a redistribution of velocity fluctuations in different directions by the pressure strain. A challenge for wall-functions is to account properly for these effects. Advanced wall-functions attempt to cope with this problem by solving the velocity profile for each wall-cell with a one-dimensional wall-normal discretization e.g. [8]. This approach shows interesting results providing accuracy without excessively compromising computational speed.

4.4 Conjugate heat transfer simulations

Most commonly, CFD simulations include only the fluid domain, while the solid surface defines the boundary. For the study of velocity fields this approach is quite convenient, since the solid surface fully defines the boundary condition (zero velocity). If the problem at hand includes a fluid-solid thermal interaction, including part of the solid in the simulation should be considered. These simulations are referred to as "conjugate heat transfer" simulations. This approach provides information not only on the fluid, but also on the solid's temperature, which is of interest in engine thermal simulations.

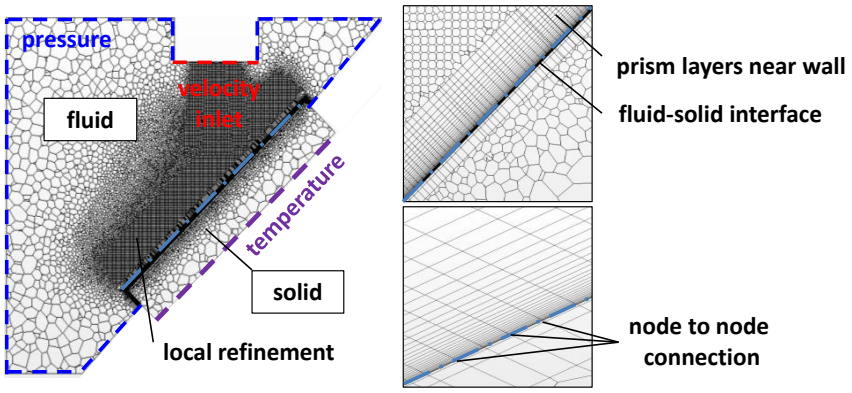

Figure 12: Example of conformal mesh with node-to-node connection at the fluid-solid interface.

Conjugate heat transfer simulations offer a number of advantages. In a transient event the temperature at the fluid-solid interface changes with time. Temperature is the boundary condition for the energy equation. Inclusion of the solid part does not "lock" the boundary value for the fluid energy equation, but leaves the possibility for it to change dynamically. This makes the system of equations converge more easily. Consequently, the overall simulation runs faster, even in spite of an increased number of cells. Indeed, for the solid domain, only the energy equation for conduction needs to be solved. This equation does not require a particularly high computational effort compared with the system of equations modeling the fluid domain as discussed in section 4.1.

A good practice to set up a conjugate heat transfer simulation is to use a meshing technique that ensures a node-to-node connection at the fluid-solid interface (see Figure 12). With such a mesh there is no interpolation error transferring information between the two different domains (fluid and solid).

4.5 Spatial discretization (mesh)

Figure 12 is an example of a mesh used in a conjugate heat transfer analysis. The example is taken from the author's work on impinging jet heat transfer on inclined surfaces. Academic cases often involve simple geometries that can be represented with a 2D domain. This is practically never the case in industry, where meshing represents a real challenge. This might be one of the biggest differences between academic and commercial applications. Modern commercial meshing tools have greatly improved the possibility to mesh with high quality even complex domains. A well-known meshing strategy is to use higher resolution in the location of highest interest and, in general, in the areas characterized by strong gradients. For example, Figure 12 shows the mesh refinement in the free jet and near wall area, including the solid domain. Areas with large gradients are not always known beforehand, when making the mesh. Some modern simulation packages offer the option of "on the run" mesh refinement based on the local gradient of a variable of choice.

4.6 Boundary conditions

A numerical simulation cannot be more accurate than the accuracy of its inputs. This sentence summarizes the basic importance of boundary conditions. Results from simulations are often compared with experimental measurements and in many applications measurements are used as input to the simulation itself. Measurements have limited accuracy and in some cases the accuracy level itself is not easy to determine. It is therefore not wise to choose a model with high accuracy if boundary conditions with similar levels of precision are not available.

5 Contributions to the field by the author

5.1 A study of single-pulse impinging jet heat transfer

Inside the engine, for every cycle, a number of rather different flow fields are established. Each flow has its specific thermal interaction with the local boundary. The cumulative result of these interactions is the net heat flux to the respective engine surfaces. Knowledge of the cumulative thermal effect is generally enough when considering the thermal load experienced by the material at some depth within the surface. Nonetheless, in some cases it is interesting to study the time-resolved heat transfer effects at the fluid-solid interface. At the solid surface there can be significant local temperature variations at each cycle. This might lead to failure by high-frequency thermal fatigue.

Of all the different flow fields occurring in the combustion chamber, the impinging jet resulting from diesel injection is identified as a good candidate to test the ability of CFD simulations to predict heat transfer in engines. Two characteristics of impinging jets lead to this choice. The first is physical: impinging jets have the highest level of heat transfer known for single phase flows. Furthermore, these effects are not uniform but vary significantly with location on the impingement surface. Secondly, impinging jet flows are quite challenging to simulate. Within the flow field there is a potential jet core and a shear layer with different turbulent characteristics. Moreover, close to the impingement surface, there is a strong interaction of the wall with wall-normal turbulent fluctuations. Most commonly used turbulence models are known to perform poorly when applied to predict these flows. It is reasonable to think that a turbulence model capable of predicting impinging jet heat transfer is likely to capture appropriately also any other flow in the engine. At least as far as the thermal fluid-solid interaction is concerned.

The premises above were the starting point for a project carried out by the author, which resulted, among other things, in the present thesis. Here the work is reported briefly with the twofold purpose to fulfill the requirements of a Ph.D. dissertation manuscript, but also to serve as a specific example of the general concepts introduced in the previous chapters. The project research question can be formulated as follows:

“To which extent is it possible to capture engine-relevant, impinging jet-like, heat transfer phenomena with high resolution numerical simulations? Furthermore, is it possible to use such simulations as a benchmark for product-development oriented simulation tools?”

In accordance with the research question, work was carried out by the author with the goal of directly comparing the performance of different CFD approaches in predicting impinging jet heat transfer. Impinging jets are a common topic in fluid mechanics research and there is an established body of literature on stationary impinging jets. A literature review can be found in the introduction of PAPER 1. In the same paper the physics of stationary impinging jets is also described. Since the main focus was on the near-wall physics, low-*Re* models were chosen for the study, in accordance with the principles treated in the previous chapters. The work resulted in a study spanning through a number of turbulence models, steady-state and transient, mesh topology and density,

near-wall mesh refinements and discretization schemes. The results were compared with multiple experiments available in literature and the work was collected in PAPER 1.

As argued in the introduction of PAPER 2, two impinging jets configurations relevant to diesel injection were found to be interesting but not exhaustively addressed in the literature: the effect of single-pulse impinging jets, and the effect of jets on inclined surfaces. In diesel engines, a jet forms at each injection and it is experienced by the piston wall as an independent pulse. Furthermore, due to the piston-top shape and its motion, the jet interacts with a surface at different angles.

The single-pulse impinging jet, as experienced in Diesel engines, is the downstream effect of a complex series of phenomena beginning with fuel injection. In the actual case the liquid fuel is injected at high pressure, undergoes primary and secondary break up, evaporates, entrains surrounding air, self-ignites and eventually collides with the piston wall. The most straightforward approach would be to find or develop a simulation including all these events. This is the object of spray and combustion research. Unfortunately, such simulations are not straightforward to implement. Furthermore, even if such simulations were available, it is extremely challenging to measure time-space resolved thermal effects in a running engine with high accuracy.

The problem was approached with a different strategy. Instead of beginning from fuel injection, following the flow downstream, it was considered to start from the wall and to move upstream towards the injector. The question to be answered then became: what type of flow could be representative of diesel injection and simultaneously be easily accessible for multiple measurements? Studying the vapor-phase velocity profile of diesel sprays, it was found that they have the features of a gas jet sufficiently downstream the nozzle [9]. This similarity has been exploited by different authors in the development of CFD models for diesel spray, e.g. [10, 11]. The “equivalent jet” approach is indeed used to estimate vapor-phase spray characteristics, such as the velocity profile.

5.2 Experimental design

A reference experiment was designed to study single-pulse impinging jet heat transfer on inclined surfaces with relevance to engine technology. The primary objective was to serve as reference for CFD simulations. Figure 13 is a graphical overview of the experimental setup and of the different measuring technologies used. A gas injector was obtained from a production GDI injector. The injector tip was modified to have a single-hole nozzle. The injector was fed with synthetic air and controlled with the engine control unit. The result was an engine-relevant jet, with respect to size, velocity profile and timing as explained in PAPER 2. The jet was pointed to a flat target with a relative positioning comparable to injector-to-piston clearance. This creates a controlled, repeatable event, which retains the fundamental characteristics of the pulse impinging jet in engines. The whole apparatus is kept at atmospheric conditions. The differences between engine environment and atmospheric conditions are mainly related to the fluid’s properties, not the flow. These differences can be accounted for with thermodynamic relations. On the other hand, the loss of relevance is believed to be well compensated by the advantage of easy access for multiple measurements.

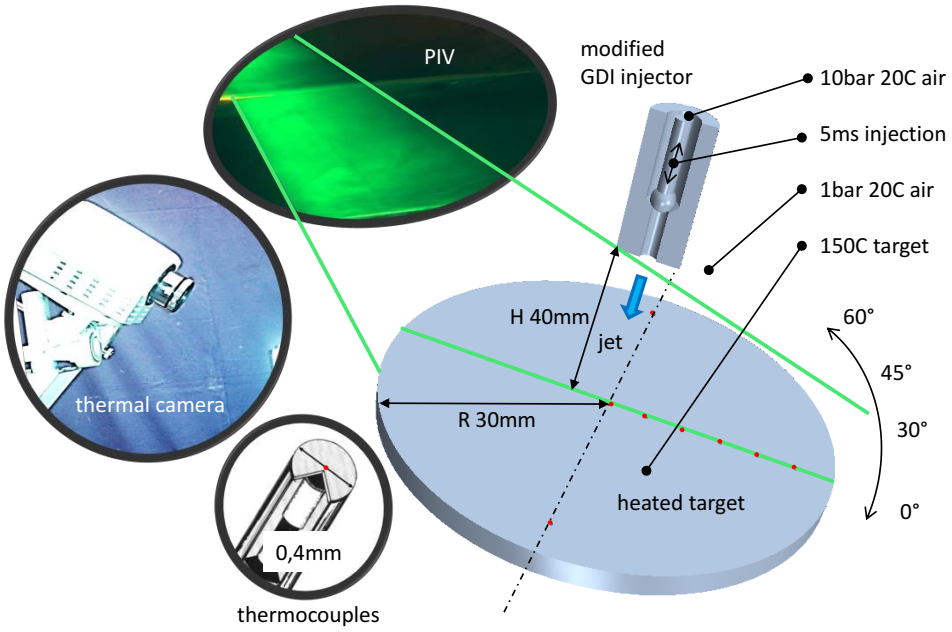

Figure 13: Experiment description.

To measure the event three different technologies were deployed (Figure 13). PIV (Particle Image Velocimetry) was used to measure the time-space evolution of the jet pulse. To measure the thermal effects it was necessary to create a temperature difference. The target was made in aluminum and fitted with heaters. The target surface temperature was measured with two methods: high rate thermocouples and infrared camera. These two methods measure the same quantity, but provide high resolution in time and space respectively.

The entire flow field for different target inclination angles was measured with PIV, providing information about the evolution of the entire event. Target surface temperature for the same configurations was also measured. The combination of these two measurements gives a picture of both the fluid and the solid side allowing the study of their interaction. The purpose of the study is indeed to replicate this in the numerical environment. Further experiments were carried out in a smaller window, focusing on the nozzle area. This provided high time-space resolution of the jet's evolution in a location suitable as inlet boundary conditions for the simulations. PAPER 3 reports the experimental results to make them available to other researchers interested in carrying out similar simulations. Particular attention is paid to present the data in a format suitable to be used as boundary conditions for simulations, both as input and for result comparison. The results are also collected in a database accessible via the internet at the following address http://www.tdf.chalmers.se/~lada/projects/jet-exp/proright.html

On the basis of the experimental setup, two numerical studies were carried out. The first study was qualitative-oriented and focused on the ability of the simulations to capture the general behavior of the jet impinging on surfaces with different angles. This work is

collected in PAPER 2. The second study focused on the ability of the simulation to accurately reproduce the experiment from a quantitative-statistical point of view. This second work is reported in PAPER 4. The summarized results from the two studies are presented in the following sections.

5.3 Single-pulse impinging jet heat transfer on inclined surface

PAPER 2 reports a campaign of simulations matching all the jet-target configurations measured. The simulations are carried out using LES and the results are qualitatively compared to the measurements. Besides validating the CFD methodology, this exercise brought to a physical insight on the nature of this phenomenon.

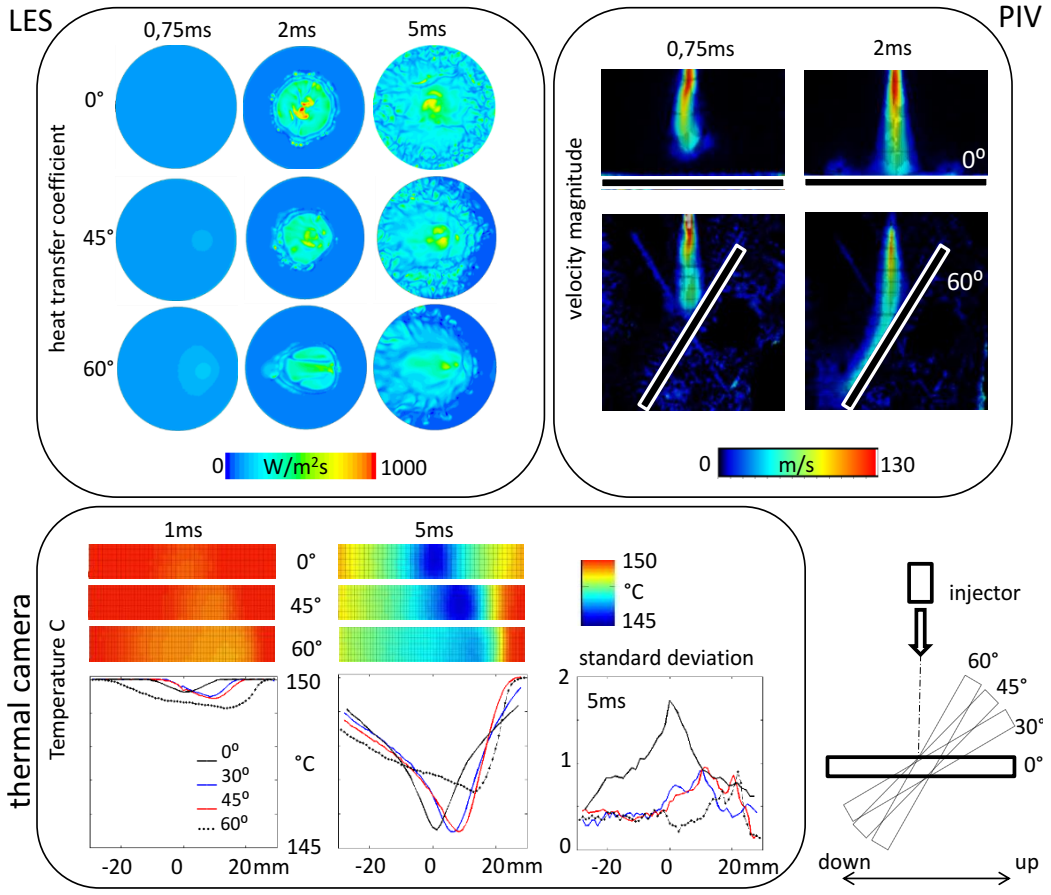

Figure 14: Single-pulse impinging jet results, compilation at different times and target inclinations. LES: instantaneous heat transfer coefficient at the fluid-solid interface. PIV: velocity magnitude average of 100 samples. Thermal Camera: surface temperature contour plot and profile along the line of maximum inclination and relative standard deviation. Average of 20 samples.

Figure 14 is a graphical summary of the results presented in PAPER 2. As experimental results, the figure reports the average values for the velocity field measured with PIV and target surface temperature. The temperature is reported both as contour plot and as profile along the line of maximum inclination. For the LES campaign the results presented are the instantaneous heat transfer coefficient on the impingement surface. This information is not available from any experiment and it gives an instantaneous picture of the thermal interaction. The surface temperature profile is the time integral effect of this instantaneous interaction. Further, the instantaneous heat transfer coefficient brings qualitative information on the turbulent structures resolved by the simulation. Results are reported for different times and target inclinations.

The results reveal that the jet penetrates faster with increasing target inclination. This can be seen directly in the velocity field but also, indirectly, on the heat transfer coefficient plots. Indeed, it can be seen that the thermal interaction begins earlier for inclined surfaces. Also the temperature measurements confirm that the thermal interaction begins earlier with inclined surfaces, but this takes some reasoning. The more the surface is inclined, the larger the change in temperature after 1ms from start of injection. This might wrongly suggest that the heat transfer is larger on an inclined surface. Further observation rules out this interpretation. Indeed, the rate of change in surface temperature is highest for the case with the jet impinging normally, as can be seen comparing the results for 1ms and 5ms. The interpretation is confirmed by the instantaneous heat transfer plots. A remarkable finding is that there is a sudden switch in thermal interaction between the two inclination angles, 45˚ and 60˚. The effect can be appreciated comparing the shape of the two temperature profile plots at 5ms. For the same angles there is a noticeable change in turbulent structures reflected on the heat transfer coefficient distribution. The good agreement upon multiple effects not only confirms the quality of the measurements but also gives confidence on the level of accuracy of the LES simulation. Even with a slight surface inclination, the fluid in front of the jet has a preferential way to escape. As a consequence, the jet travels significantly faster. Another consequence is on the jet's repeatability. This can be shown with statistical analysis of surface temperature measurements. Indeed, the standard deviation for the jet approaching a normal surface is significantly higher than any other case, as shown in the results.

5.4 Replicating the experiment using LES

The numerical study was extended to achieve a quantitative-statistic comparison between with the experimental results. The results are reported in PAPER 4. A LES model was used to accurately replicate the experiment and it was run multiple times to allow for statistical analysis of the results.

Figure 15 summarizes the results of this investigation. In general a high level of agreement is found between the numerical simulation and the experimental results. The figure compares characteristics for both the flow and the thermal interaction. The velocity profile is compared 1/3 of the distance between the simulation inlet and the target, in the free-jet region. Target temperature evolution is followed showing good agreement in magnitude and distribution, for both the absolute values and their standard deviation. In PAPER 4 the results are compared more extensively, including also the early stages of

the jet. Furthermore, in the aforementioned paper, the possible reasons for discrepancies are discussed.

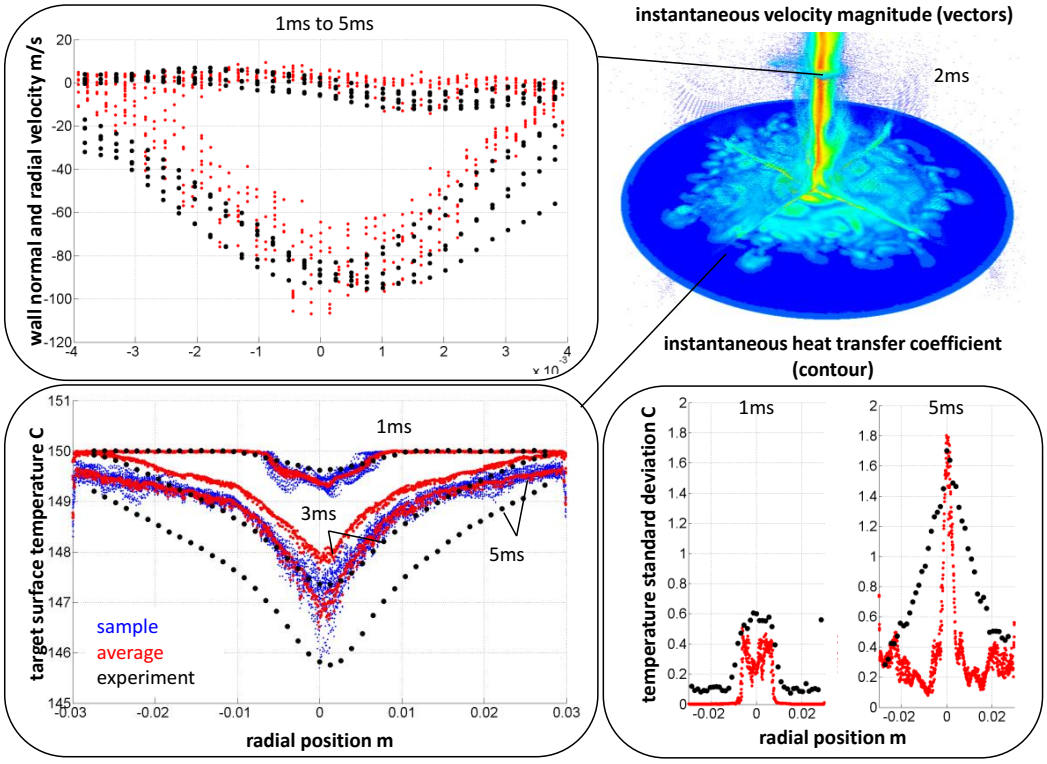

Figure 15: Quantitative-statistical comparison between LES and experimental results. Experimental values average over 100 samples for velocity and 20 samples for temperature. LES average over 10 samples. Scatter plot of all computational cells corresponding to the measurement locations.

5.5 Considerations behind the approach

The work reported in this chapter can be used as an example to make general considerations on how to approach the study of complex-flow heat transfer. An easily underestimated strength of the approach presented here is the possibility to compare results for the very same event obtained from completely independent sources. Using side-by-side experimental and numerical methods it is possible to gain deep insights on the phenomenon, which would be otherwise not readily available. One of the specific ways this can be done is in the interpretations of dubious results. Cross-checking results obtained with different methods allows to corroborate them or rule them out: achieving this distinction would prove rather difficult without this process.

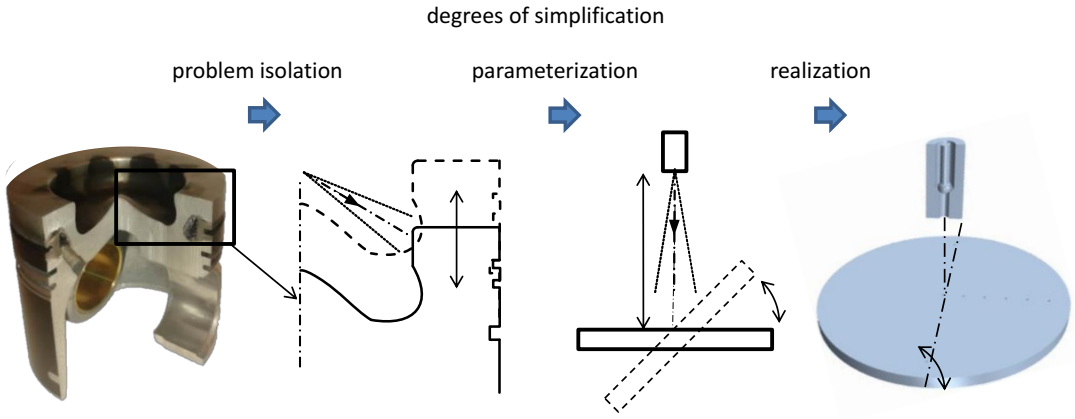

Figure 16: Degrees of simplification.

Figure 16 gives a graphical interpretation of the simplification process behind this work. A major obstacle when attempting to measure processes in engines is the complexity of the system and the hostile environment in the combustion chamber. A large part of basic engine research is not carried out in real engines, but in facilities which offer better access for measurements (e.g. spray bombs, optical engines, etc.). The experiment presented above was designed with the following constraints in order of priority.

1. Be as simple as possible and achievable with the available resources (time, costs)
2. Generate information well suited as a CFD reference
3. Be relevant to the diesel injection process

This prioritization might appear to be wrongly stated, having the constraint of "relevance to diesel injection" placed last. However, prioritizing the closeness to the real event would have rapidly increased complexity. With complexity follows uncertainty, which in turn affects the trustworthiness of the entire study. Instead, by keeping things as simple as possible, it was easier to measure and simulate the phenomenon with higher accuracy and multiple techniques. Results based on such an approach are founded on more solid grounds and can be used as milestones for the study of more complex systems.

5.6 Conclusions regarding the author's work

If considered alone, this work brings little, if any, immediate improvement on the practical front. The CFD simulations employed are still too computationally expensive to be implemented in the nearest future. Nevertheless, the research question is answered and the work defines a modeling methodology capable to reproduce impinging jet heat transfer with a reasonably high degree of accuracy. Furthermore, simulations carried out with this method are believed to be of sufficient reliability to serve as a benchmark for other models. Indeed, this work was carried out within the frame of a larger project. This project includes the implementation and testing of CFD simulations suitable for product development (URANS with wall-functions).

Another outcome of the present work is the new, simple way to create a gas injector. In this study the modified GDI injector is used to produce a repeatable single-pulse

impinging jet to study fluid-solid thermal interaction. Other researchers are currently using this technique to create injectors for natural gas and hydrogen. These researches are oriented towards gas-direct-injection engine technology and are focusing, among other things, on the jet-tip air entrainment.

6 Concluding remarks

This work summarizes the basic principles behind heat transfer in engines. More specifically, it focuses on the thermal interaction between the thermodynamic cycle and the surfaces containing it. These principles are the foundations for creating numerical simulations capable of capturing this phenomenon. The modeling approach is treated as well, making this work a potential reference for engineers interested in simulating heat transfer in engines.

The first step towards successful modeling is the understanding of the physical laws governing the event of interest. This work begins with a chapter presenting the principles of heat transfer. Convective heat transfer plays a major role in engine heat losses. Furthermore, it is directly associated with specific problems such as the failure of Diesel pistons due to thermal stresses. Understanding convective heat transfer requires considerations on the interaction between the turbulent flow and its solid boundaries. This interaction is often treated focusing on the velocity field. This work, instead, discusses also the interaction with heat transfer along with the classic description.

Most features of engine flows are unique for this technology. Chapter 3 in this work gives an overview of the different flow patterns that develop in the engine at different locations and at different phases of the engine cycle. Each of the described flows can be analyzed in far more detail. The purpose here is to raise awareness on the complexity of the system. The description attempts to connect the flows to their relative heat transfer effects. Chapter 3 also introduces the specific case of impinging jet-like flows in Diesel engines, which was the object of specific research by the author.

Numerical methods are a welcome modern tool in engine development. They allow to develop the technology further, and faster. For a successful implementation of a numerical analysis it is necessary to combine two aspects: a sufficient knowledge of the problem under investigation and good understanding of modeling. Lack of either can easily bring results with no factual value. Chapter 4 in this work explains the basics of CFD modeling. A variety of methods exists, and selecting the most appropriate one to investigate a specific problem is very important. This choice is facilitated by having a good understanding of the methods, the concepts and the working principles. This work wishes to provide the background necessary to confidently approach the challenge of simulating heat transfer in engines.

The last chapter presents the author's research in the field of impinging jet heat transfer. Specifically, the research focuses on the ability of CFD simulations to capture impinging jet heat transfer phenomena in engine-relevant configurations. The work carried out might be seen as an exercise to gain knowledge and confidence regarding numerical methods for the prediction of heat transfer in complex flows. The research also brought some insights on the physics of pulse impinging jet itself. These notions are possibly of more interest for the scientific community studying this specific flow, rather than for the engineering community developing engines. Therefore, these findings are collected in the author's publications here appended. The research work is briefly reported in the text to serve as an example to discuss a method to approach complex technological challenges.

Beginning from the real effect of jet impingement in Diesel engines, it provides a case-study well suited for the comparison of experimental and numerical investigations. The approach is not confined to this specific case and might be taken as an inspiration for other future studies.

References

1. ESA 2000 Annual Meeting DAB: April 6, 2000
2. J.B. Heywood, “Internal Combustion Engines Fundamentals”, ISBN 0-07-028637-X ,1988
3. J. Bredberg, “On the Wall Boundary Condition for Turbulence Models”, Dept. Thermo and Fluid Dynamics, Chalmers University of Technology, int. report 00/4, 2000
4. A. Alharbi, “high-speed high-resolution vector field measurements and analysis of boundary layer flows in an internal combustion engine” PhD Thesis, University of Michigan, 2010
5. J. Dec, "A Conceptual Model of DI Diesel Combustion Based on Laser-Sheet Imaging", SAE Technical Paper 970873, 1997
6. J. Eismark, “On soot oxidation in heavy duty diesel engine combustion” Licentiate thesis ISSN 1652-8565, 2012
7. T. Husberg, S. Gjiria and I. Denbratt, “Piston Temperature Measurements by use of thermographic phosphors and thermocouples in a heavy-duty diesel engine run under partially premixed conditions”, SAE 2005-01-1646
8. T. J. Crafta, S. E. Ganta, H. Iacovidesa & B. E. Laundera, “A new wall function strategy for complex turbulent flows”, Numerical Heat Transfer, Part B: Fundamentals: An International Journal of Computation and Methodology, Volume 45, Issue 4, 2004
9. D. L. Siebers. “Scaling liquid-phase fuel penetration in diesel based on mixing-limited vaporization” SAE 1999-01-0528
10. N. Abani, R. D. Reitz, “Modeling sub grid scale mixing of vapor in diesel spray using jet theory”, ICLASS 2009, 11th Triennial International Annual Conference on Liquid Atomization and Spray Systems, Vail, Colorado USA
11. S. Wook Park, R. D. Reitz, “A gas jet superposition model for CFD modeling of group-hole nozzle sprays”, Volume 30, Issue 6, 1193–1201, 2009