Creation and destruction of in-cylinder flows; Large eddy simulations of the intake and the compression strokes

by

Martin Söder

March 2015
Technical Reports from
Royal Institute of Technology
KTH Mechanics
SE-100 44 Stockholm, Sweden

©Martin Söder
Universitetsservice US–AB, Stockholm 2015
Abstract
The aim of this thesis is to increase engine efficiency by studying the flow structures created in an engine cylinder during the intake phase and the effect of the subsequent compression.

The invention of the combustion engine has enabled three centuries of economic growth fueled by energy stored as hydrocarbons. However, during the latter part of the twentieth century negative consequences on health and environment of the combustion engine were observed. In order to reduce emissions without increasing fuel consumption, improved knowledge of all physical processes occurring in the engine are necessary. The aim of this thesis is to increase the understanding of the flow prior to combustion, which can lead to reduced engine emissions and fuel consumption.

Intake flow structures are studied using large eddy simulations and experiments on a steady swirl test rig. Flow acceleration was observed to reduce the swirl coefficient, and higher swirl coefficient was found during valve closing as compared to during valve opening. This implies that the rotation is stronger during the later part of the intake then what has been previously assumed. In addition, the computations show that the volume above the valves has a profound effect on the swirl created during the intake. To take this into account a novel way of calculating the swirl number was suggested. This approach gives a lower swirl number as compared to the commonly used Thien methodology. The effects of compression are studied using simulations of predefined flow structures undergoing compression. The peak turbulence levels were found to be increasing with tumble number and decreasing with swirl. It was noted that compression increased the turbulent fluctuations in the cylinder axis leading to anisotropic turbulence and that a small tilt angle was observed to have a significant effect on swirl homogeneity at top dead center. In this thesis, a new methodology was proposed and validated for calculation of in-cylinder turbulence for a flat piston.

The results of the thesis enhance the understanding of the dynamic effects encountered during intake as well recognizing that a small tumble component has a strong effect on the flow structures prior to combustion. These results can be used to improve the simplified computational methods used to optimize the engine.

Descriptors: engine turbulence, engine simulations, swirl, tumble, intake flow structures, compression.
Sammanfattning  
Målet med denna avhandling är att möjliggöra renare och bränslesnålare motorer genom att studera luftflödet i en motor under insugs- och kompressionsfasen. 
Ekonomisk tillväxt och ökat välstånd har till stor del skett tack vare förbränningsmotorn. Förbränning av kolväten leder dock till utsläpp av bland annat växthusgaser och kväveoxider. För att minimera dessa måste förståelsen av de fysikaliska processerna i motorn före och under förbränning ökas.


I denna avhandling har en ny metodik utvecklats, validerats och används för att utvärdera turbulensen under kompression. Denna avhandling ger dels en bättre förståelse för de dynamiska effekterna under insugsfasen. 

Nyckelord: motor, turbulens, large eddy simulation, swirl, snurr, tumble.
Preface

This doctoral thesis in fluid mechanics is based on investigations of the flow during intake and compression using primarily numerical simulations. The intake stroke is mainly examined using a so-called swirl test rig with different boundary conditions, while the compression stroke is studied using a cylinder undergoing compression with different initial conditions. A novel methodology to estimate turbulent kinetic energy is presented, validated and used. The thesis is divided into two parts in where the first part, starting with an introductory essay, is an overview and summary of the contribution to the field of engine design. The second part consists of 7 papers. In chapter 12 of the first part in the thesis the respondent’s contribution to all papers are stated. The work presented in this thesis has been performed by the author under supervision of Dr. Lisa Prahl Wittberg and Professor Laszlo Fuchs.

March 2015, Stockholm

Martin Söder
Ignorance more frequently begets confidence than does knowledge.

Charles Darwin
## Contents

Abstract iii

Sammanfattning iv

Preface v

Chapter 1. Introduction 1
  1.1. Open questions/aim of thesis work 2
  1.2. Contributions 3

Chapter 2. Internal combustion engine 5
  2.1. Combustion processes 5
  2.2. Emissions from diesel combustion 7
  2.3. Emission Legislation 8
  2.4. The Combustion Engine; History, energy and the economy 10

Chapter 3. Flow structures of internal combustion engines 14
  3.1. Swirl and tumble 14
  3.2. Tumble breakdown 16
  3.3. Squish flow 17
  3.4. In-cylinder Turbulence 18
  3.5. Ideal and real flow structures in commercial engines 19
  3.6. Simulation and averaging 22

Chapter 4. Models and methods 23
  4.1. Governing equations 23
  4.2. Particle Image Velocimetry, PIV 24
  4.3. Computational Fluid Dynamics 25

Chapter 5. Evaluation methods 35
  5.1. Vorticity 35
5.2. Kinetic energy 36
5.3. The swirl number and swirl coefficient 38
5.4. Turbulent anisotropy 41
5.5. Turbulent length and time scales 42

Chapter 6. Flow under consideration 43
6.1. Dynamic effects during the intake 43
6.2. Effects of compression 48
6.3. Full cycle simulation 52

Chapter 7. Validation 54
7.1. Grid study, intake 54
7.2. Grid study, compression 56
7.3. Validation of methodology 57

Chapter 8. Results and discussion 58
8.1. Results, Dynamic effects during intake 58
8.2. Results, Effects of compression 68
8.3. Results, Piston motion 82

Chapter 9. Summary and conclusions 87
9.1. Dynamic effects during intake 87
9.2. Effects of compression 88
9.3. Piston motion 89
9.4. Suggested implementation of the results during engine design 89
9.5. Correct swirl number 89
9.6. Effect of non-zero tilt 90
9.7. Implications for SI engines 90

Chapter 10. Self criticism and model uncertainties 91
10.1. General comments 91
10.2. Dynamic effects during intake 92
10.3. Effect of compression 93
10.4. Piston motion 94

Chapter 11. Proposed future work 96

Chapter 12. Papers and authors contributions 98

Acknowledgements 101
References

Paper 1. A Coupled PIV-LES Approach to Understand Port Generated Structures 114

Paper 2. Compression of a swirling and tumbling flow 134

Paper 3. Effects of compression on coherent structures in an enclosure 156

Paper 4. Study of flow generated by the port in a heavy-duty diesel engine at different valve lifts using PIV 172

Paper 5. Effect of swirl/tumble (Tilt) angle on flow homogeneity, turbulence and mixing properties 188

Paper 6. Towards understanding the effect of compression on swirl and tumble in the context of turbulence and mixing 214

Paper 7. Investigating the dynamic effects on flow structures generated during the intake stroke in heavy-duty diesel engines using Large Eddy Simulations 234
Part I

Overview and summary
CHAPTER 1

Introduction

_Beware of the person of one book._
Thomas Aquinas

_The book you don’t read won’t help._
Jim Rohn

The internal combustion engine has been a major part of the society for more than 150 years. It has drastically increased human prosperity and is today vital for our society to function. Nevertheless, the combustion process has some non-negligible negative side effects such as emitting local and global pollutants. Vehicle emissions have shown to pose a great health hazard in many cities, among others Los Angeles that due to its location is especially prone for smog. Smog is a serious health hazard, EPA (1998), and in order to reduce smog emission legislation has been introduced. In Europe the so-called Euro legislation started in 1992 with the Euro I legislation regulating nitrogen oxides, NO\textsubscript{x}, and particle mass per kilowatt-hour of work. The regulation has since become stricter and from January 2014, all new trucks have to conform to the Euro VI legislation.

Different strategies can be used to meet these emission targets. Examples of such are injection timing, Selective Catalytic Reduction (SCR) and Exhaust Gas Recirculation (EGR) to reduce nitrogen oxide emissions as well as filters and increased injection pressure to reduce particle mass. Generally, these strategies lead to increased fuel consumption and/or cost. Thus, a trade-off between local emissions (NO\textsubscript{x} and particles) and global emissions (Carbon dioxide, CO\textsubscript{2}) exist. For the transport sector, lower fuel consumption is equivalent to lower cost. Therefore, engine manufactures seek to minimize fuel consumption for a given emission level. This is the main motivation for this thesis. Moreover, fuel consumption is tightly linked to carbon dioxide emissions. Hence, reducing fuel consumption will increase profits and reduce carbon dioxide emissions per transported goods. Additionally, as conventional crude oil production peaked in 2006, IEA (2010), further increases in oil production rely on more expensive unconventional sources, oil prices are probably to remain
high and volatile. Due to high oil prices and an increasing concern of climate change the effort of reducing fuel consumption will most likely increase for both economic and political reasons.

In order to reduce fuel consumption without increasing emissions, improved knowledge of all physical processes occurring in the engine are necessary. The work presented in this thesis aim at increasing the understanding of the flow prior to combustion, i.e. the intake and compression strokes. Increased understanding of the pre-combustion flow will help the engine manufactures to design an intake geometry that will provide a flow field (rotation and turbulence) and fluid composition (mixing of exhaust gases and intake air) at the start of ignition (SOI) such that the combustion process is optimized. In this work, the intake flow is studied using a so-called steady swirl test rig geometry with fixed and changing boundary conditions and valve lifts to improve the understanding of the dynamic effects during intake. The described geometry has been studied in detail for stationary flows. However, with novel boundary conditions, previously unknown effects of acceleration, valve motion and geometry have been found. The effects of compression were studied using simplified engine geometry with innovative initial conditions. The initial conditions were used to study the effect of a tilted rotational motion undergoing compression as well as the effects of swirl on turbulence. Compression was found to amplify axial fluctuations while turbulence was damped by rotation around the cylinder axis. In order to validate the methodology with which turbulence was calculated several cycles of a motored engine were simulated. These simulations were also used to introduce and validate an improved methodology to calculate the mean swirl number.

1.1. Open questions/aim of thesis work

... as we know, there are known knowns; there are things we know we know. We also know there are known unknowns; that is to say we know there are some things we do not know. But there are also unknown unknowns - the ones we don’t know we don’t know.

Donald Rumsfeld

Generation of in-cylinder flow structures

Currently, the swirl number is measured under stationary conditions in swirl test rigs. The aim of this work was to scrutinize the assumptions behind the measurements, which are:

1. The flow into the cylinder is adiabatic and incompressible and thus proportional to the piston displacement (100 % volumetric efficiency)
2. The angular momentum is conserved (no viscous losses).
3. The flow is Reynolds number independent, the same flow structures are created regardless of the flow velocity.
4. The flow structures are independent on whether the piston accelerates or decelerates.
5. The velocity of the flow passing the valves is much greater than the speed of the valves. The flow structures are therefore not affected by valve motion.
6. The created flow structures are only dependent on valve lift (consequence of 4 and 5).

The effect of compression

Several studies have been done on either tumble breakdown or the effect of swirl on emission formation. There are also experimental studies that indicate a combination of swirl and tumble may be beneficial, Lee et al. (2007); Hill & Zhang (1994); Gosman (1985); Furuno et al. (1990). However, since these are experimental studies it is difficult to know if this is due to a change in flow kinetic energy or due to a change in rotational tilt. Therefore, studying how mixing, turbulence and flow structures at TDC are affected by rotational tilt and strength is of great importance. Additionally, the physical understanding of tumble breakdown is not complete and further insight is still of importance.

1.2. Contributions

Study hard what interests you the most in the most undis-\v
\nuplined, irreverent and original manner possible.

Richard P. Feynman

The work presented aim at increasing the understanding of dynamic effects during intake and compression. Some of the main findings presented in this thesis that will contribute to the field of engine development/research are:

- Non-stationary flow over the valves was found to have an effect on swirl. The swirl coefficient was greater during flow acceleration (early intake) compared to during flow deceleration (late intake).
- The swirl coefficient is higher during valve closing as compared to the same valve lift during valve opening due to inertia.
- During the time it takes for the volume above the valves to enter the cylinder no angular moment around the cylinder axis enters the cylinder. This leads to a lower swirl number inside the engine compared to what can be expected from steady swirl test rig measurements. A new methodology for calculating swirl that considers this was proposed.
1. INTRODUCTION

- At low valve lifts the root-mean-square of the swirl coefficient fluctuations is greater than the mean swirl coefficient.
- Swirl has a dampening effect on turbulence.
- Compression increases axial fluctuations (anisotropic turbulence).
- A small deviation from pure swirl leads to a non-uniform rotation at TDC (speed-up in the cylinder center).
- Peak turbulence levels increases with the tumble number at BDC.
- TDC turbulence level was found to be relatively insensitive to tumble or tilt.
- A new methodology for estimating turbulence for a flat piston was proposed and validated.
- Engine speed was observed to have a moderate (increasing) effect on in-cylinder fluid mass and tumble number.
CHAPTER 2

Internal combustion engine

The main principle of the internal combustion engine is to convert chemical energy into mechanical work. There are several ways this can be done, but the most commonly used engine cycle is the four-stroke engine. The *four-stroke engine* works in four distinct, although sometimes overlapping, phases:

**Intake stroke** (or induction stroke) The piston starts at top dead center (TDC), moving downward while the intake valves are opening. This process draws air through the intake ports pass the valves into the cylinder. The geometry of the intake ports directs the air, creating large-scale in-cylinder motions. Around the end of the intake phase, close to bottom dead center (BDC), the valves close.

**Compression stroke** During the compression stroke, the piston moves upward, compressing the air. The ratio between the volumes at BDC and TDC is called the compression ratio. The compression stroke ends with the piston reaches firing TDC (fTDC) and **Combustion** is initiated. During combustion, the chemical energy of the fuel is converted into potential energy in the form of pressure.

**Power stroke** As the piston moves downward, the potential energy is converted into mechanical work. The power stroke is the only phase that performs work, while the other phases require work supplied by the other cylinders.

**Exhaust stroke** When the exhaust valves open around BDC, exhaust gases are first evacuated by the pressure difference between the cylinder and exhaust manifold, i.e. the blowdown phase. Once the pressure difference has been reduced, the remainder of the exhaust gas is forced out from the cylinder by the piston. This is called scavenging.

### 2.1. Combustion processes

For internal combustion engines, two different types of four-stroke combustion processes are traditionally used; spark ignited (SI) and compression ignited (CI) combustion. The former is used in gasoline engines, where the fuel is mixed with the air prior to TDC when a spark ignites the mixture. In the latter, the compression ignition engine, the fuel is injected close to TDC and the heat
caused by the compression ignites the fuel. This type of engine, use primarily diesel fuel and is often called diesel engines. In a spark ignited engines the fuel is mixed with the air at start of combustion (SOC). The combustion process is thus entirely premixed. In the direct injected diesel engine, several processes occur before and after the main diffusion combustion, see Fig. 2.1.

**Figure 2.1.** Example of in-cylinder pressure and heat release rate showing combustion processes in a diesel engine.

*Ignition delay.* After SOI, it takes a small amount of time for the fuel spray to start burning (SOC). This time is known as ignition delay and is caused by the time it takes for the first fuel droplets to evaporate, mix with air and reach combustion temperature. When combustion temperature has been reached, a small amount of the fuel is mixed, leading to premixed combustion, Heywood (1988).

*Premixed combustion.* During premixed combustion, a thin flame front travels with the laminar flame speed. However, the combustion process is expedited by turbulence that moves the flame front by turbulent convection. By increasing the turbulence levels, the characteristic eddy turnover time to laminar burning time is reduced. This in turn leads to increased turbulent flame speed, reducing the combustion time, Bradley *et al.* (1992).

*Diffusion-flame.* Immediately after the premixed fuel has been burned, the rate of heat release is governed by the fuel injection rate. This part of the
2.2. EMISSIONS FROM DIESEL COMBUSTION

Diffusion-flame or mixing-controlled combustion is known as diffusion-flame or mixing-controlled combustion and is when most chemical energy is released. In a diffusion flame, the combustion process is limited by the air fuel mixing/diffusion. Generally, this leads to slower burning flames with higher soot levels than for premixed combustion.

Post-oxidation. After fuel injection has ended, 20-45% of the chemical energy of the fuel remains as soot. Oxidation of this soot is important both from an environmental and an economical point of view. Post-oxidation has traditionally been intensified by swirl leading to lower soot emissions, see e.g. Benajes et al. (2004); Dembinski (2013).

2.2. Emissions from diesel combustion

Engine emissions can be divided into two subcategories; local and global. Local emissions such as NO\(_x\), HC, SO\(_2\), CO and particles, have negative impact on the close surroundings leading to health issues. Global emissions on the other hand affect the entire globe. Legislation has so far focused on local emissions, which in turn has forced the engine developers to focus on these emissions as well. Global emissions, mainly CO\(_2\), is tightly linked with fuel efficiency and is thus minimized for economic purposes. In the following text, the (main) emissions that form due to combustion of diesel fuel are listed and briefly discussed. For a more thorough description of the processes, see e.g. Heywood (1988).

Nitrogen oxides. At high temperatures nitrogen oxides, NO and NO\(_2\), commonly referred to as NO\(_x\), form through the extended Zeldovich mechanism (Thermal NO\(_x\)). For direct injected diesel engines, high temperature zones (and thus production of NO\(_x\)), are located in the interface between the fuel spray and the compressed air, see Dec (1997). Nitrogen oxides is harmful to the lungs at high concentration and contribute to acid rain as well as the formation of smog, EPA (1998). NO\(_x\) is the main local emission from modern diesel engines.

Particles. The second major emission from modern diesel engines is particles. Particles are suspected to increase the risk for lung cancer, and have been shown to increase respiratory symptoms, Jungnelius & Svartengren (2000). Particle matter is created in areas where the air-fuel mixture is rich at relatively low temperatures. Legislation has reduced the allowed limit of particle matter in the exhaust drastically. In the Euro VI legislation, the total number of particles is targeted as well as the total mass, discussed further in Sec. 2.3.

Unburned hydrocarbons. In regions where the fuel for different reasons does not burn completely, due to too low temperatures and/or a lack of oxygen, unburned hydrocarbons, HC, remain. Emitted into the atmosphere, HC is a health hazard and can produce smog. Diesel engines generally produce low amounts of HC.
**Sulfur dioxide.** Fossil fuels contain different amounts of sulfur, which can be removed to a large extent during refining. However, in some countries high quantities of sulfur is still present in the fuel, Wikipedia (2013). During combustion sulfur binds with oxygen, creating sulfur dioxide, $\text{SO}_2$, that mixed with water will create sulfuric acid leading to acid rain. Sulfuric acid also has the effect of being highly corrosive, leading to higher engine wear. Reduction of $\text{SO}_2$ emissions is mainly achieved by reducing the sulfur content in the fuel.

**Carbon monoxide.** As an intermediate step in the combustion between HC and carbon dioxide, carbon monoxide, CO, is formed. The oxidation of CO to $\text{CO}_2$ mainly occurs with the help of OH-radicals. This process requires oxygen and temperatures above $1200 \text{ K}$. Interruption of the process can be caused by local lack of oxygen or low temperature. CO is very toxic to humans and can cause death as it blocks the ability of blood to bind to oxygen, Raub *et al.* (2000).

**Water.** One of the major emissions from combustion engines is water vapor. However, it is not considered a pollutant.

**Carbon dioxide.** In all combustion processes including hydrocarbons, most of the fuel is converted to water and carbon dioxide, $\text{CO}_2$. Carbon dioxide is inert, not harmful to animals and vital to plants. However, carbon dioxide is known to be a greenhouse gas, already identified as such by Arrhenius (1896). It is generally accepted that it has a non-negligible impact on the global climate, IPCC (2007). Emissions of $\text{CO}_2$ and the atmospheric $\text{CO}_2$ level can be seen in Fig. 2.2 and Fig. 2.3, respectively. Depending on model the effect on the climate if the emissions of greenhouse gases is not halted range from bad to extremely bad, Pierrehumbert (2014).

**Nitrous oxide.** Laughing gas or nitrous oxide, $\text{N}_2\text{O}$, is a 300 times more potent greenhouse gas as compared to carbon dioxide. It is not formed during combustion but can be formed in the selective catalytic reduction (SCR) catalysts used to reduce the emission of nitrogen oxides. Depending on catalytic design, the emission of nitrous oxide differs significantly.

### 2.3. Emission Legislation

In Europe, heavy-duty vehicles must conform to the European emission regulation framework. The first regulation framework was introduced in 1992 and called Euro I. Since 1992, the allowed level of pollutants has been significantly reduced, see Fig. 2.4. Since 2014, all new engines in the European Union must meet the Euro VI regulations.

#### 2.3.1. Future legislation

As a direct cause of the introduction of stricter emission legislation, the improvements in combustion efficiency have been postponed. Temperature is
2.3. EMISSION LEGISLATION

Figure 2.2. World carbon dioxide emission (Production-based accounting), data: World Bank (2011).

Figure 2.3. Atmospheric CO$_2$ level measured in ice cores obtained at the Law Dome, East Antarctica Etheridge et al. (1998) and atmospheric measurements at Mauna Loa Observatory, Hawaii NOAA/ESRL & Tans (2015).

namically important in all combustion related chemical processes. In order to obtain maximum fuel efficiency, and low CO$_2$ emissions, all available fuel should combust at TDC. However, if obtained, this would lead to temperatures where formation of NO$_x$ is very fast. Between the introduction of the Euro I and Euro V legislation (1992-2009), the global emissions of carbon dioxide increased with 43 %, see Fig. 2.2. The combined effect of combustion on atmospheric CO$_2$ is shown in Fig. 2.3, suggesting that carbon dioxide emissions are likely to be
implemented in future legislation, Euro VII+. Additionally, emissions of nitrous oxides are an avoidable side effect of SCR catalysts. Therefore, emission legislation regarding nitrous oxides may also be enforced in the future.

2.4. The Combustion Engine; History, energy and the economy

All economic activity requires energy. Obvious but an often overlooked fact when the role of the combustion engine in society is analyzed. The coupling between available energy and economic growth was understood by early economists when land was the primary source of energy through photosynthesis, Smith (1776). In this context, the industrial revolution and the following economic growth can be viewed as a drastic increase in available energy. Fig. 2.5 shows how the invention of the combustion engine spurred a drastic increase of economic growth, fueled by stored hydrocarbons; coal and later oil. The increase in energy consumption and composition is depicted in Fig. 2.6. The work produced by the combustion engine led to a drastic increase in standard of living. However, as the price of energy (work) fell with the industrial revolution other factors became limiting to economic growth. Therefore, most economists started to believe that energy is of lesser importance, and exponential economic growth can be independent on energy. The belief between a decoupling of land/energy and the economy is well stated by the libertarian think tank Adam Smith Institute, Butler (2001):

"Today we see no limit to economic growth. Our capital and technology give rise to all kinds of new business sectors and opportunities for employment. In Smith’s time, however, the economy was dominated by agriculture, and he mistakenly sees the impossibility of developing land beyond its fertility as a limit
For the transporting sector, keeping fuel cost low is necessary for profitability and growth. Since conventional crude (cheap) oil production peaked in 2006, IEA (2010), the increase in oil production has been forced to rely mainly in unconventional oil production in the US (shale oil), see Fig. 2.7. This has lead to higher extraction costs, Barclays Capital estimate that the upstream capital expenditure (CAPEX) has increased by 10.9 % per annum in the period 1999-2013 (from $5 to $22/bbl), see Kopits (2014). Oil producing countries have become dependent to high oil prices to avoid fiscal deficits, e.g. Saudi Arabia and Russia requires a oil price of $78/bbl and $116/bbl, respectively, Lewis (2012) (Deutche Bank). For OPEC an output-weighted break-even price is $105/bbl, APICORP (2011).

Benes et al. (2014) estimates a near doubling (from $100/bbl) of the oil price in the coming decade, yet arguing that their model is not pessimistic from a geological view. Kopits (2014) estimate a crude oil price of $150/bbl by 2020 and argues that the demand driven forecasting (first estimate economic growth then calculate needed oil production) commonly used (IEA, EIA...) should be replaced with a combined demand/supply driven forecasting model. Goldman Sachs, see Kopits (2014), estimate that an oil price above $100/bbl is necessary for the oil industry to have positive cash flow after capital investment and dividends. Regardless of these forecasts/estimates the oil price has fallen significantly since July 2014. This fall in oil price coincide with the increased production of shale oil in the US and has reduced investment in new oil fields. EIA (2015) estimates that the oil production in the US will start to decline in May 2015, which in turn lead to higher oil prices. Therefore, high and volatile oil prices will most likely remain in the future. From this perspective, there is a high possibility that fuel economy and diversity will be even higher prioritized in the coming decade.
Figure 2.5. 2000 years of gross world production growth with some historical landmarks, data: DeLong (1998) and for 2000-2011 World Bank (2011).
2.4. THE COMBUSTION ENGINE; HISTORY, ENERGY AND THE ECONOMY

Figure 2.6. World Energy consumption, source: BP (2013) and Grübler (1999).

Figure 2.7. Oil production, source: BP (2013).
CHAPTER 3

Flow structures of internal combustion engines

The flow structures found in the cylinder of an internal combustion engine are characterized by swirl, tumble, squish and small-scale turbulence. During the intake phase, the fluid enters the cylinder through the valves, forming jets. The jets induce angular momentum forming coherent structures such as swirl and tumble, see Sec. 3.1. Some of the jet energy will be converted into turbulence and in the early to mid-intake stroke turbulence levels will be very high, Lumley (1999). In the second half of the intake stroke, turbulent production is significantly reduced as the intake jet vanishes. This in turn leads to a rapid decay of small-scale turbulence and by the end of the intake stroke only low levels of turbulence are found, Celik et al. (2005).

During compression, the increase in density and the changes in length scales (due to geometrical change) have an amplifying effect of the remainder of the turbulence, Lumley (1999). Moreover, the reduction of geometrical length scales affects both tumble and turbulence level, Sec. 3.1. For pistons with a piston bowl an inward fluid motion will be introduced at the end of the compression stroke known as squish, see Sec. 3.3.

3.1. Swirl and tumble

Engine swirl number, SN, is defined as the gas angular velocity around the cylinder vertical center axis, $\Omega_{\text{Swirl}}$, normalized by the angular velocity of the crank shaft, $\Omega_E$, Eqn. (3.1) see Heywood (1988). Tumble number (TN) is defined as the gas angular velocity around an axis perpendicular to swirl, $\Omega_{\text{Tumble}}$, passing the cylinder center of gravity, Eqn. (3.2). A more thorough description of how swirl and tumble numbers are calculated can be found in Sec. 5.3

$$SN = \frac{\Omega_{\text{Swirl}}}{\Omega_E} \tag{3.1}$$

$$TN = \frac{\Omega_{\text{Tumble}}}{\Omega_E} \tag{3.2}$$

The strength of the different motions can be chosen by careful intake port design. If a swirling motion is wanted, the flow from the intake ports should be directed in the tangential direction of the cylinder. Tumble can be created
3.1. SWIRL AND TUMBLE

in a similar manner by adjusting the angle to the cylinder axis at which the flow enters the cylinder.

Both swirl and tumble are large-scale structures that dissipate slowly. Subjected to compression, swirl and tumble will be affected differently. As the angular momentum around the cylinder axis is conserved during compression (if viscosity losses are small), the evolution of swirl depends solely on moment of inertia around this axis. For a piston with a bowl, the moment of inertia around the cylinder axis is decreased (mass is directed inward) and swirl is increased during the late part of compression. The tumble motion is affected by both a change in angular momentum (will be discussed in the next section) and a decrease in moment of inertia. The decrease of moment of inertia will act in order to increase the tumble angular velocity.

Hall & Bracco (1987) noticed that the swirl number was approximately constant with engine speed. However, Lioi & Santavicca (1983) observed a decrease in swirl number with engine speed. This discrepancy is likely caused by engine breathing capacity, Hill & Zhang (1994). Although there are exceptions, Daimler (2011), most CI engines exhibit a swirling gas motion, as swirl has been found to reduce soot emissions in CI engines, Jayakumar et al. (2012); Benajes et al. (2004); Dembinski & Ångström (2013). The reduction of soot is caused by two effects; firstly, by deflecting the fuel spray and thus hindering it from reaching the wall; secondly, by increasing the post-oxidation of the soot. However, an increase of swirl increases heat transfer to the cylinder walls, Hill & Zhang (1994); Woschni (1967). An increase of swirl has also been linked to an increase of premixed combustion leading to higher peak combustion temperatures and consequently higher thermal thermal NO\textsubscript{x}, Benajes et al. (2004); Dembinski (2013). Jayakumar et al. (2012) also found that increasing the swirl number from a modest 1.44 to a very high 7.12 increased the particle number in the exhaust. According to Vermorel et al. (2009) low tumble numbers increase cycle-to-cycle variations for SI engines, which is consistent to what was reported by Fogleman et al. (2004).

Tilt angle

The ratio between swirl and tumble has been defined using the angle between the axis of rotation and the cylinder axis, in this thesis referred to as the tilt angle, $\alpha$, see Fig. 3.1. Zero and ninety degree tilt angle is equivalent to pure swirl and tumble, respectively.
3.2. Tumble breakdown

When a tumbling motion is compressed, based on the conservation of momentum and the reduction in moment of inertia, it can be argued that the tumble number should increase. However, this is only true at the beginning of the compression stroke. Around mid-stroke it has been observed that the tumble motion breaks down with subsequent rapid increase of small-scale turbulence, see e.g. Lumley (2001). Müller et al. (2009) found, using planar particle image velocimetry (PIV) on a SI engine, that tumble breakdown occurs around 30 crank angles before TDC and that peak total kinetic energy is shifted towards earlier crank angles with increasing engine speed. Arcoumanis et al. (1990); Fogleman et al. (2004) found tumble breakdown between 60 to 30 crank angles before TDC while Gosman et al. (1985); Haworth et al. (1990) saw tumble breakdown 30 crank angles before TDC. Although this feature is less difficult to apprehend (sharp reduction of available geometry), the actual mechanism and onset of the breakdown has long been argued. Lumley (2001) discussed work by Obukhov who showed that a solid body type of rotation is unstable around the second axis of inertia, arguing that this is applicable to a tumbling flow. According to Lundgren & Mansour (1995), conversion of an elliptic vortex into turbulence is caused by elliptic instabilities, causing a wave on the primary vortex, distorting the vorticity into a thin vortex layer on which secondary instabilities grow, see Kerswell (2002).

In order to study the effect of compression on tumble, several numerical and experimental studies have been made on a well-defined tumbling flow in a rectangular geometry at a compression ratio of four at 206 rpm and with a tumble number of approximately 8. Borée et al. (2002) noticed that at a volumetric ratio of two, the mean kinetic energy was transferred from the tumble motion to turbulence at a timescale in the order of the vortex turnover time. It was also noticed that small deviations in the large-scale motion (tumble) was amplified during compression. Moreover, Borée et al. (2002) has linked the instability to elliptical as well as centrifugal instabilities, it has also been found that the
3.3. SQUISH FLOW

If the piston has a bowl, this will at the end of compression force the flow into the piston bowl, see Fig. 3.2. This inward facing flow is called squish and is responsible for an increase in swirl (spin-up) close to TDC and the opposite in the beginning of the expansion phase due to the change of moment of inertia around the cylinder axis. Depending on swirl number, centrifugal forces will affect the squish flow differently. For low swirl numbers, the squish flow will follow the cylinder head. For high swirl numbers the flow will be forced down, following the piston bowl creating a vortex rotating in the opposite direction as observed for low swirl numbers, see Miles (2009) and Fig. 3.2. This is explained by the higher centrifugal force for the high swirl number case.

**Figure 3.2.** Illustration of squish flow. Low and high swirl, respectively.
3. FLOW STRUCTURES OF INTERNAL COMBUSTION ENGINES

3.4. In-cylinder Turbulence

Turbulence is usually defined as the fluctuation ($\vec{u}'$) about the mean velocity ($\vec{U}$), Eqn. (3.3). Thereafter turbulent kinetic energy ($tke$) is calculated using Eqn. (3.4).

\[
\vec{u}' = \vec{u} - \vec{U} \quad (3.3)
\]
\[
tke = 0.5\overline{\vec{u}' \cdot \vec{u}'} \quad (3.4)
\]

where $\vec{U}$ is the mean velocity and $\overline{\vec{u}' \cdot \vec{u}'}$ is the mean of the dot product $\vec{u}' \cdot \vec{u}'$. However, no clear definition of in-cylinder turbulence exists, as the observed value will be highly influenced by the averaging procedure of the mean flow. The choice of averaging procedure should therefore be dependent on flow conditions. For in-cylinder flows all averaging procedures have different drawbacks, the most common procedures are:

**Time-Averaging**

Time averaging is the most commonly used method for statistically stationary flow. However, in-cylinder flow is not stationary due to the movement of the valves and piston. If there is a large-scale separation between the turbulent scales and that of the piston motion, it can be reasonable to decompose the flow into a slowly evolving mean with high-frequency fluctuations. In this case, unsteady RANS simulations may be appropriate. The RANS methodology is described in Sec. 4.3.1.

**Spatial-Averaging**

For homogeneous turbulence, a spatial average is the most appropriate average. In-cylinder turbulence is not homogeneous, however, it can be argued that the smallest turbulent scales are independent of the larger scales. Therefore, modeling the smallest scales as carried out for large eddy simulations is a reasonable approach, see Sec. 4.3.2. However, the turbulence levels will, if a spatial average is used depend on the spatial domain. Thus, the kinetic energy of the larger turbulent structures will not be included in the estimate of turbulent kinetic energy.

**Phase-Averaging**

Phase-averaging is in many ways the most appropriate averaging method for in-cylinder flows, as the cycle mean and deviations from the mean becomes clearly visible. However, phase-averaging does not make a distinction between large-scale cycle-to-cycle variations and turbulence. Moreover, information about individual cycles and corresponding mixing properties are lost. The estimated turbulence level using a phase-averaged velocity is likely to be an overestimation of the "true" turbulence due to cycle-to-cycle variations, Glover *et al.* (1988).
Energy based method

In this work, an energy based method to quantify the turbulent kinetic energy has been used. In this method, all structures that are not quantifiable (i.e. not caused directly by the piston motion or are of rotating type) are described as turbulence. This method and principle thereof will be described in more detail in Sec. 5.2.

Measured in-cylinder turbulence

Integral length scales within the engine at TDC has been found to be in the order of 6-10 mm for an SI engine, Li et al. (2002). Lee et al. (2007) found that maximum turbulence at TDC was obtained for a tilted rotating motion (both swirl and tumble) with a tilt angle of 61°. In-cylinder turbulence levels at TDC have been found to scale with engine speed, Liou & Santavicca (1983); Liou et al. (1984). Baritaud (1989) found that the turbulence intensity $u'_{rms}/MPS$, where $MPS$ is the mean piston speed and $u'_{rms}$ is the root-mean-square of the turbulent velocity fluctuations, at TDC for SI engines is in the order of 0.6 while Liou & Santavicca (1983) found turbulence intensity in the range of 0.4 and 0.6.

3.5. Ideal and real flow structures in commercial engines

Although the structures differ between CI and SI engines the main motivation for both engine types is to create a flow at SOC that influence the combustion process, e.g. increasing combustion rate, increasing post-oxidation, reducing soot production or reducing peak combustion temperatures.

3.5.1. CI engines

Generally, the fuel injector is located in the center of the cylinder with several holes creating several fuel sprays. Therefore, it is likely that the optimal flow at SOI is circumferentially homogeneous. The peak combustion temperatures are dependent on the fuel spray. Thus, optimal flow and EGR distribution may not necessarily be radially uniform. The optimal swirl level on the other hand will likely depend on injection pressure, piston bowl and exhaust aftertreatment, and is therefore a compromise during engine design.

3.5.2. SI engines

Turbulence intensity and length scale have a profound influence on combustion for SI engines, Enaux et al. (2011a); Nishiyama et al. (2014). Generally, soot formation is lower for SI engines as compared to CI engines. Therefore, gas/fuel mixing, in-cylinder turbulence levels and total kinetic energy prior to the spark are of greater importance as compared to a swirling motion, Buschbeck et al. (2012); Jackson et al. (1997). In order to have low cycle-to-cycle variations (CCV) it is important that the flow close to the spark plug is of similar strength.
and direction for the different cycles, Hill & Zhang (1994). A tumbling motion has been found to give favorable in-cylinder conditions at SOC for SI engines, Lumley (2001).

3.5.3. Measurement of swirl and tumble

Swirl number is usually measured under steady conditions in a so-called swirl test rig. The measurements are carried out for different valve lifts using a honeycomb torque meter, Tippelmann (1977), or a turbine flow meter, Thien (1965). Flow of angular momentum can be obtained by measuring the torque around the cylinder axis on a honeycomb. For paddle wheel measurements it is necessary to assume solid body rotation and a homogeneous axial velocity profile to estimate the flow of angular momentum. For the paddle wheel, additionally to constant axial velocity a solid body type rotation is assumed. Tumble number is more difficult to measure, an estimate can be obtained by measuring the torque vector on a honeycomb and not only around the cylinder axis. A characteristic swirl and tumble number is calculated by weighting the numbers for different valve lifts. The weight of the different valve lifts is dependent on the amount of mass that can be assumed to enter the cylinder for each of the lifts, see Sec. 5.3 for equations. Hence, similar numbers can be obtained by qualitatively different flows, Rezaei et al. (2010). Generally, the measured swirl number is an overestimation as compared to the actual swirl number, Crnojevic et al. (1999).

Swirl measurements are done regularly by engine manufactures and have been reported by Bensler et al. (2002); Kaario et al. (2007) and Vernet (2012) among others. Bensler et al. (2002) hypothesized that minimizing the swirl fluctuations measured in the swirl test rig would reduce cycle-to-cycle variations. Jackson et al. (1997) looked at seven different port designs to correlate different flow characteristics with combustion performance, using PIV in a steady flow rig. For spark ignition engines, the most important flow characteristics were found to be tumble and overall flow kinetic energy. Historically, steady swirl test rig measurements have worked relatively well to characterize the flow inside a diesel engine, but with the increasing demand, deeper knowledge of the created structures is needed.

3.5.4. Swirl in sector models

Currently, three approaches of simulating in-cylinder flow and combustion are used. The most computationally heavy is Large Eddy Simulations (LES) and is mainly used in academic papers to understand basic mechanisms of the flow, e.g. Bottone et al. (2012); Haworth & Jansen (2000); Wu & Perng (2002); Enaux et al. (2011b); Rezaei et al. (2010); Toledo et al. (2007); Celik et al. (2005); Vernorel et al. (2009); Adomeit et al. (2007); Richard et al. (2007); Liu & Haworth (2011). However, due to faster simulation time k-ε Reynolds
Averaged Navier Stokes (RANS) models are de facto industry standard for computing turbulent engine flow, Miles (2009). It is well known that the k-\( \varepsilon \) model (and other models based on Boussinesq’s isotropic eddy viscosity assumption) has problems in rotating flows, Versteeg & Malalasekera (2007). These models are also unable to capture the three dimensional breakdown of tumble, Le Roy & Le Penven (1998). Nevertheless, RANS simulations are able to give valuable insights into in-cylinder flows and help during engine design, Payri et al. (2004); Ge et al. (2008); Yamato et al. (2001); Adomeit et al. (2006); Dembinski & Ångström (2013). For diesel engines that are dominated by a rotating fluid motion around the cylinder axis (swirl) the rotational symmetry can be used by only simulating a sector of the combustion chamber. The low computational time for a RANS sector simulation makes it possible to simulate a large number of different geometric and combustion setups, e.g. Thundil Karuppa Raj & Manimaran (2012); Bonatesta et al. (2007); Ge et al. (2010); Wickman et al. (2001, 2003).

The use of sector models has made correct measurements of the swirl number more important. If the swirl number differs between what is measured in a steady swirl test rig (and used in the sector model) and the real engine, a sub-optimal combustion system is obtained. One major drawback of sector models is that rotational tilt is neglected, rotational tilt and not only swirl number, as will be shown in Sec. 8.2, has an influence on the flow field at TDC. Therefore, possible implications of tilt on the results need to be assessed and discussed when results from a sector model are presented.

3.5.5. Flow studies of complete engine cycles

There are several numerical and experimental studies that have studied the flow in realistic engine designs. Haworth & Jansen (2000) were among the first to conduct large eddy simulations on an engine geometry. It was noticed that a defined flow, i.e. clear tumble, reduced cycle-to-cycle variations. Buschbeck et al. (2012) noted, using PIV measurements on an SI engine, that the total flow kinetic energy is very important to reduce cycle-to-cycle variation at stoichiometric fuel-air mixtures. For lean operating conditions it is observed that the large-scale motions close to the spark plug is more important than total kinetic energy. Similarly, Enaux et al. (2011a) noticed that the variation in velocity close to the spark plug is the key driver of cycle-to-cycle variations for SI-engines. Although very important in the late part of compression Payri et al. (2004) noted, using RANS and LDV, that the piston bowl shape has a very marginal effect on the creation of structures during intake. Yamato et al. (2001); Li et al. (2006) found that a stratified fuel-air mixture is possible in a pure tumble gasoline engine, using RANS simulations and PIV measurements.

Studies of the effect of off-centered swirl on combustion include Dembinski & Ångström (2012), who noticed using Combustion Image Velocimetry
(CIV, i.e. PIV using soot particles) experiments, that off-centered swirl sur-
vives compression and has an effect on combustion and post-oxidation. Similar
observations have been done by Adomeit et al. (2006), who observed, using
RANS simulations and experiments, that soot formation is strongly influenced
(increased) by inhomogeneities of the swirl motion. Rezaei et al. (2010) no-
ticed using LES that eccentricity in the in-cylinder swirl pattern affects soot
production and oxidation, it was also noticed that different valve lift strategies
resulting in the same measured swirl number will impact combustion differently,
more specifically an increased inhomogeneity leads to higher soot emissions.

3.6. Simulation and averaging
Moureau et al. (2004) noted that first order mapping between different meshes
reduces the accuracy of the simulations. Enaux et al. (2011b) noticed that
at least 25 cycles were necessary to get accurate mean values close to TDC,
to obtain converged RMS values (less than 10 % error) at least 50 cycles are
needed. It was also noted that the traditional Smagorinsky subgrid scale (SGS)
model with low order (2nd) schemes are too dissipative. Celik et al. (2005)
reviewed the literature of LES for internal combustion engines. It was reported
that even coarse grids were able to capture interesting features. Moreover,
turbulence created during intake decays quickly and only low turbulence is
detected in the early part of the compression stroke. Numerical grids with
large aspect ratio (i.e. the ratio between the cell widths in different directions)
were reported to reduce the accuracy of the simulation significantly.
CHAPTER 4

Models and methods

*In theory, there is no difference between theory and practice.*

*But in practice, there is.*

Yogi Berra

In order to obtain relevant results, different approximations have been done concerning numerics. In this chapter the different methods and models used throughout this thesis are presented. The flow is governed by the Navier-Stokes equations which are presented first. This is followed by the different methods (LES and PIV) used to obtain the results.

In Sec. 4.1 and 4.3.2 equations are written in tensor notation using Einstein summation convention. Equations are otherwise written in vector form.

4.1. Governing equations

The fluid flow inside an engine cylinder is governed by the compressible Navier-Stokes equations:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \quad (4.1)
\]

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \tau_{ij} \quad (4.2)
\]

\[
\frac{\partial \rho e_0}{\partial t} + \frac{\partial}{\partial x_j} [\rho u_j e_0 + u_j p + q_j - u_i \tau_{ij}] = 0 \quad (4.3)
\]

where \( \rho \) is the density, \( u_i \) is the velocity vector in the \( i \)-direction, \( p \) is the static pressure, \( \tau_{ij} \) is the viscous stress tensor, \( e_0 \) is the total energy defined as \( e_0 = e_I + \frac{1}{2} \rho u^2 \), \( e_I \) is the internal energy and \( q_i \) is the heat flux. For a Newtonian fluid the viscous stresses are proportional to the strain rate and viscosity, \( \mu \):

\[
\tau_{ij} = 2\mu (S_{ij} - \frac{1}{3} S_{kk} \delta_{ij}) \quad (4.4)
\]

where \( S_{ij} \) is the strain rate tensor defined as:

\[
S_{ij} = \frac{1}{2} (\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) \quad (4.5)
\]
In order to close the equations an equation of state is needed. It is provided by the ideal gas law:

\[ p = \rho RT \]  \hspace{1cm} (4.6)

where \( R \) is the specific gas constant and \( T \) is the temperature.

Moreover, due to the large temperature increase during compression, the viscosity cannot be assumed constant. To compute the viscosity as a function of temperature, Sutherland’s law has been used:

\[ \mu = \mu_{ref} \left( \frac{T}{T_{ref}} \right)^{3/2} \frac{T_{ref} + S}{T + S} \]  \hspace{1cm} (4.7)

where \( \mu_{ref} \) is the viscosity at the reference temperature \( (1.716 \times 10^{-5} \text{ kg/ms}) \), \( T_{ref} \) is the reference temperature \( (273.15 \text{ K}) \) and \( S \) is the Sutherland temperature \( (110.4 \text{ K}) \).

4.1.1. Passive scalars

In some of the work presented in this thesis passive scalars have been used. A passive scalar is governed by the advection-diffusion equation. For a scalar, \( C \), it can be written as:

\[ \frac{\partial \rho C}{\partial t} + \nabla \cdot (\rho \overrightarrow{u} C) = \nabla \cdot (\rho D \nabla C) \]  \hspace{1cm} (4.8)

where \( D \) is the molecular diffusivity. If the Schmidt number defined as:

\[ Sc = \frac{\mu}{\rho D} \]  \hspace{1cm} (4.9)

is assumed constant and assuming that the viscosity is constant in space, Eqn. (4.8) can be written as:

\[ \frac{\partial \rho C}{\partial t} + \nabla \cdot (\rho \overrightarrow{u} C) = \frac{\mu}{Sc} \nabla^2 C \]  \hspace{1cm} (4.10)

In the presented work the Schmidt number has been assumed to have a value of 0.9. However, mixing for the presented cases is likely to be driven by advection (turbulence), molecular diffusion should therefore have a secondary (negligible) effect.

4.2. Particle Image Velocimetry, PIV

Parts of the results presented in this thesis were obtained using PIV measurements on a steady flow test rig. The PIV methodology will be described briefly in the next sections, for a more extensive review see Raffel et al. (2007).

The basic principle behind PIV is rather simple; seed particles are introduced into the flow, from two subsequent images, the distance traveled by the
4.3. COMPUTATIONAL FLUID DYNAMICS

particles between the two snapshots are measured using correlation. Thereafter, by dividing the distance by the time between the images, the velocity field is obtained.

However, applying PIV in practice is more challenging. The most basic PIV measurement is the 2D-PIV, typically consisting of one camera and one light source (laser). The laser illuminates the particles twice and the camera records the images. The images are then divided into interrogation windows. For each of these windows, the correlation between the two images at different displacements, of the second image, is carried out until a peak is achieved that will provide the velocity field. Spurious errors do however occur and in order to minimize these, different algorithms have been developed, see e.g. Westerweel (1994).

2D-PIV only provides two velocity components in a plane. By adding a second camera, the third velocity component can be obtained, commonly referred to as stereo-PIV. To have the image, lens and object plane to intersect, Scheimpflug adapters are necessary, see Prasad & Jensen (1995). However, this introduces perspective distortions since the magnification factor differs across the field of view. The different methodologies available to treat these images can be found in literature, see Prasad (2000).

4.2.1. Difficulties

In order for the seeding particles to correctly follow the flow, the particles need to be small and of the same density as the surrounding medium. Any deviations will contribute to the measurement error. Additionally, too long time between images will lead to particles passing the light sheet, and no correlation will be obtained between images. On the other hand, too short interval will result in a signal-to-noise ratio that is insufficient. This is a major problem in areas with large velocity differences, such as around the valves. A more practical problem encountered in this study was due to the reflections around the glass cylinder arising from the incoming laser. Reflections from the glass cylinder are an order of magnitude greater than that of the particles studied; this is problematic as the imaging cameras can be damaged by light with too high intensity. Reducing the possible power output from the laser and thus reducing the visibility of the particles.

4.3. Computational Fluid Dynamics

In this section some common approaches to model the turbulence is described first. This is followed by the numerical algorithm used and different discretization schemes. Finally, the standard computational setup used in the simulations is presented in Sec. 4.3.5.
In order to solve the Navier-Stokes equations correctly, all scales of the fluid flow must be resolved. This is called Direct Numerical Simulation (DNS). However, this methodology is computationally very expensive and not feasible for (most) industrial applications, modeling is thus necessary. Traditionally, this has been carried out through the Reynolds Averaged Navier-Stokes (RANS) equations. Lately, the increase in computational power has made large eddy simulations attractive for more complex geometries.

4.3.1. Reynolds Averaged Navier-Stokes, RANS

The RANS equations are obtained by decomposing the flow into a mean and a fluctuating component and thereafter the equations are averaged, see any basic fluid dynamics book e.g. Versteeg & Malalasekera (2007); Pope (2008); Anderson (1995). The decomposition will result in a new set of terms, containing the effect on the mean flow by the turbulent fluctuations. In order to close the new set of equations, model equations have to be applied. The most common turbulence model is the two equation $k-\epsilon$ model. Although widely used, the $k-\epsilon$ model has well-known deficiencies, where poor performance with some unconfined flows such as axisymmetric jets, flows with large strains, e.g. curved boundary layers, and rotating flows. These shortcomings are general for models based on Boussinesq’s isentropic eddy viscosity assumption, see e.g. Versteeg & Malalasekera (2007). As the in-cylinder flow of heavy-duty diesel engines includes swirling motions, any isotropic eddy viscosity assumption will not describe the physics correctly. However, from an engineering point, this approach has proven to be very useful during engine design.

4.3.2. LES

Different from RANS simulations were all turbulent structures are modeled, large eddy simulations only models the smallest scales. The motivation behind this is that for high enough Reynolds numbers, the small-scale turbulence only affects the larger fluid scales in a dissipative manner via the energy cascade. This simplifies the modeling and makes it more universal, see e.g. Pope (2008). According to Sagaut (2001) the energy transfer between two modes (different wave numbers) separated more than two decades are negligible.

The incompressible LES equations are obtained from the incompressible mass and momentum equations:

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{4.11}
\]

\[
\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} (u_i u_j) = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \tag{4.12}
\]
4.3. COMPUTATIONAL FLUID DYNAMICS

By filtering the Navier-Stokes equations using a low pass filter, $\tilde{\cdot}$, the incompressible LES equations are obtained as:

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0 \quad (4.13)$$

$$\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} = -1 \frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \right] + \tilde{\sigma}_{ij} \quad (4.14)$$

where $\tilde{\sigma}_{ij}$ is the SubGrid Scale (SGS) stress tensor:

$$\tilde{\sigma}_{ij} = \tilde{u}_i \tilde{u}_j - \tilde{u}_i \tilde{u}_j \quad (4.15)$$

In order to solve the equations the SGS stress tensor $\tilde{\sigma}_{ij}$ has to be modeled. Several models exist but in this thesis the traditional Smagorinsky as well as the implicit model are presented.

4.3.2a. Modeling of the subgrid scale stress tensor. Modeling of the SGS is done by a set of hypotheses, Sagaut (2001). The most commonly used hypotheses are presented below. First, the unresolved scales are hypothesized to be turbulent:

- If subgrid scales exist then the flow is locally (in space and time) turbulent

The second hypothesis leads to the class of functional models as compared to structural models, see Sagaut (2001).

- The action of the subgrid scales on the resolved scales is essentially an energetic action. The balance of the energy transfers between the two scale ranges is thus sufficient to describe the action of the subgrid scales.

- The subgrid scales have a purely dissipative effect on the resolved scales.

The second hypothesis above implies that only forward cascade is allowed. Models concerning backward energy cascade exist. However, these models are rarely used and will not be covered in this thesis. In order to close the equations hypotheses regarding the nature of dissipation are necessary.

- The energy transfer mechanism from resolved to subgrid scales is analogous to the molecular mechanisms represented by the diffusion term, in which viscosity appears.

- Characteristic length ($l_0$) and time ($t_0$) scales are sufficient for describing the subgrid scales.

By applying the above hypotheses, the following expression of the SGS stress tensor is obtained:

$$\tilde{\sigma}_{ij} - \frac{1}{3} \tilde{\sigma}_{kk} \delta_{ij} = \sigma^d_{ij} = -2 \nu_T \tilde{S}_{ij} \quad (4.16)$$

$$\nu_T \propto \frac{\nu}{l_0} \quad (4.17)$$
where \( \nu_T \) is the SGS viscosity. The isotropic residual tensor, \( \tilde{\sigma}_{kk} \), is added to the pressure leading to the modified pressure:

\[
\Pi = \tilde{p} + \frac{1}{3} \tilde{\sigma}_{kk}
\]  

(4.18)

The eddy viscosity incompressible LES equations are thus:

\[
\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \Pi}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\nu + \nu_T) \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \right]
\]  

(4.19)

4.3.2b. Smagorinsky model of the subgrid scale stress tensor. From Eqn. (4.17) the Smagorinsky model, Smagorinsky et al. (1965), takes the following form:

\[
\nu_T = (C_s \Delta)^2 \left( \langle S_{ij} \rangle \right)^{3/2}
\]  

(4.20)

where, \( \Delta \), is the filter length usually approximated with the cell volume, \( C_s \) is the Smagorinsky coefficient and \( \langle \rangle \) designates a statistical average. By using Kolmogorov’s hypothesis for isotropic homogenous turbulence:

\[
E(k) = K_0 \langle \epsilon \rangle^{2/3} k^{-5/3}, K_0 \sim 1.5
\]  

(4.21)

and assuming that the filter width is within the inertial subrange and the approximation proposed by Lilly (1966) \( \langle |S_{ij}|^{3/2} \rangle \simeq \langle |S_{ij}| \rangle^{3/2} \). The following estimation of \( C_s \) is obtained, Pope (2008):

\[
C_s = \frac{1}{\pi} \left( \frac{2}{3K_0} \right)^{3/4} \approx 0.17
\]  

(4.22)

However, the modeling constant, \( C_s \), is in practice adjusted to improve results and values ranging between 0.1 and 0.2 appear in the literature, Sagaut (2001).

4.3.2c. Implicit LES.

Whenever you find yourself on the side of the majority, it is time to pause and reflect.

Mark Twain

All practical schemes/grids introduce dissipation. The subgrid scales can therefore be modeled implicitly by assuming that the dissipative effects of the numerical schemes will affect the larger flow scales in a similar manner as an explicit scheme, i.e. the action of the subgrid scales on the resolved scales are purely dissipative. For a second order central differencing scheme this can be achieved by adding a first order upwind (UW) scheme. Due to stability reasons, this is done for most complex geometries. This effect can be illustrated using a
convective flow, \( u_i \equiv u(x_i) > 0 \), on an uniformly spaced grid, \( \Delta x = x_i - x_{i-1} \), at grid point \( i \).

\[
\left( \frac{du}{dx} \right)_i \approx \left( \frac{du}{dx} \right)_{i,W} = \frac{u_{i+1} - u_{i-1}}{2\Delta x} - \frac{\Delta x}{2} \frac{u_{i+1} - 2u_i + u_{i-1}}{\Delta x^2} \\
\approx \left( \frac{du}{dx} \right)_{i,CD} - \frac{\Delta x}{2} \left( \frac{d^2 u}{dx^2} \right)_i
\]

(4.23)

where CD stands for central difference scheme. The one-dimensional convective equation can now be written as:

\[
u_i \left( \frac{d^2 u}{dx^2} \right)_i \approx u_i \left( \frac{d^2 u}{dx^2} \right)_{i,CD} - \nu_i \left( \frac{d^2 u}{dx^2} \right)_i
\]

(4.24)

where \( \nu_i = u_i \Delta x / 2 > 0 \) is the numerical viscosity, compare with Eqn. 4.20.

If the filter cut-off length is inside the inertial subrange, the exact value of viscosity is of lesser importance as the equilibrium subrange is independent of viscosity. Furthermore, several studies show that this methodology provides accurate and in some specific cases better results as compared to explicit SGS models, see e.g. Patnaik et al. (2003); Aspden et al. (2008); Grinstein et al. (2007), or Pope (2008, p.631-634). For flows undergoing compression, there are even more compelling reasons not to use an explicit model. Subgrid-scale models, as the previously mentioned Smagorinsky model, are based on the assumption that the smallest turbulent scales are isotropic, which is the case for most fluid flows. But for flows as studied in this work including compression in one direction, anisotropy is present in the vorticity equation, Sec. 5.1. Thus, the turbulence cannot be assumed isotropic even at the smallest scales. For a flow with increasing enstrophy \( \frac{1}{2} (\omega \cdot \omega) \), is can be observed that the energy spectra do not follow the \(-5/3\) law, see Orlandi & Pirozzoli (2010) and Sec. 3.2. However, as long as most of the kinetic energy is resolved the effect should be limited regardless of whether a SGS model is used or not.

### 4.3.3. Pressure-velocity coupling, the PISO-algorithm

When solving the Navier-Stokes equations one of the most complex tasks is how to handle pressure. As can be seen in the momentum equation, Eqn. 4.2, if the pressure gradient is known, calculating the velocity field is rather straightforward. Generally, this is not the case whereby the pressure needs to be calculated as well. Since there is no explicit transport equation for pressure, the pressure is calculated using the continuity equation, Eqn.4.1, and the equation of state, Eqn. 4.6. However, at low Mach numbers this coupling is weak which leads to numerical problems.

In order to overcome this problem, different procedures based on the pressure correction equation have been developed. One of the most commonly used algorithms for transient flows is the PISO algorithm, described as follows:
4. MODELS AND METHODS

1. Predictor step, solve the discretized momentum equations.
2. Corrector step, solve the pressure correction equation (obtained by taking the divergence of the discretized momentum equations and using the discretized continuity equation) and correct pressure and velocity.
3. Second corrector step, solve the pressure correction equation using the corrected pressure and velocity.
4. Additional corrector steps can be added but according to Issa (1986), two corrector steps should be sufficient for most cases.

4.3.4. Discretization/Numerical schemes

In fluid dynamics, the advection-diffusion equation is of uttermost importance, and can be written as:

\[
\frac{\partial \phi}{\partial t} + \nabla \cdot (\vec{u} \phi) = \nabla \cdot (D \nabla \phi) \tag{4.25}
\]

where \(\phi\) is the variable of interest, \(u_i\) the velocity and \(D\) is the diffusivity. If Eqn. (4.25) is integrated over a control volume, \(V\), and Gauss’ theorem is applied Eqn. (4.26) is obtained.

\[
\int_V \frac{\partial \phi}{\partial t} dV + \int_S (\vec{u} \phi) \cdot \vec{n} dS = \int_S D \nabla \phi \cdot \vec{n} dS \tag{4.26}
\]

where \(S\) is the surface of the control volume and \(\vec{n}\) is the surface normal vector. The equation consists of three terms; a temporal term, an advective and a diffusive flux. The spatial terms will be treated first followed by the temporal term.

4.3.4a. Discretization of spatial terms. If the volume is discretized with a finite number of faces the following is obtained:

\[
\int_S \phi \vec{u} \cdot \vec{n} dS \approx \sum \phi_c \vec{u}_c \cdot \vec{n}_c \partial S_c \tag{4.27}
\]

and

\[
\int_S (\rho D \nabla \phi) \cdot \vec{n} dS \approx \sum (\rho D \nabla \phi_c) \cdot \vec{n}_c \partial S_c \tag{4.28}
\]

where the index \(c\) denotes values on the middle of the surfaces. In order to solve these equations the advective, \(\vec{u}_c \phi_c\), and diffusive, \((\nabla \phi_c)\), fluxes need to be approximated. Generally the diffusive flux, \((\nabla \phi_c)\), is approximated using a second order central difference. Contrary to the advective terms this does not pose any problems, Versteeg & Malalasekera (2007), and will thus not be further covered.

In the following \(\vec{u}_c\) is assumed to be known, in practice it is calculated separately in a similar manner. The approximation of \(\phi_c\) is illustrated for the one-dimensional case in Fig. 4.1. A simple method to approximate \(\phi_c\) would...
be a linear approximation:

\[ \phi_e \approx \phi_p + \frac{\phi_E - \phi_P}{x_E - x_P} \delta x_{pe} \]  

(4.29)

However, central differencing of the advective term, will give rise to unphysical oscillations in areas of sharp gradients. Therefore, a more stable approach is the upwind discretization:

\[ \phi_e \approx \phi_P \]  

(4.30)

The upwind scheme is stable but also diffusive. In order to overcome these shortcomings, Total Variation Decreasing (TVD) schemes have been developed, Sec. 4.3.4b.

It can be noted that all schemes can be written on the following form:

\[ \phi_e \approx \phi_P + \frac{1}{2}(\phi_E - \phi_P)\psi \]  

(4.31)

where \( \psi = \psi(r) \) and \( r \) is:

\[ r = \frac{\phi_P - \phi_W}{\phi_E - \phi_P} \]  

(4.32)

This is the ratio between the upwind-side gradient and the downwind-side gradient and can be thought of as a second order extension of the original UD scheme. For some popular schemes, the value of \( \psi \) is:

- Upwind Differencing (UD) \( \psi(r) = 0 \)
- Central Differencing (CD) \( \psi(r) = 1 \)
- Linear Upwind Differencing (LUD) \( \psi(r) = r \)
- Quadratic Upwind Differencing (QUICK) \( \psi(r) = \frac{3+2r}{4} \)

4.3.4b. Total Variation Decreasing (TVD). In order for a scheme to be TVD, the local minima must be non-decreasing and local maxima must be non-increasing, i.e. the Total Variation (TV) must not increase, see Eqn. (4.33) and Fig. 4.2, hence the name.

\[ TV = |\phi_2 - \phi_1| + |\phi_3 - \phi_2| + |\phi_4 - \phi_3| + |\phi_5 - \phi_4| \]  

(4.33)
Sweby (1984) has given necessary and sufficient conditions for a scheme to be TVD. In a $r - \psi$ relationship it can be summarized as:

- If $0 < r < 1$ for TVD schemes $\psi(r) \leq 2r$
- If $r \geq 1$ for TVD schemes $\psi(r) \leq 2$

It can be noted that:

- If $0 \leq r \leq 1$ for TVD schemes $\psi(r) \leq 2r$
- If $r$ for TVD schemes $\psi(r) \leq 2$

- the UD scheme is TVD
- the LUD scheme is TVD for $r \leq 2$
- the CD scheme is TVD for $r > 0.5$
- the QUICK scheme is TVD for $\frac{3}{7} \leq r \leq 5$

This implies that in order to make a scheme TVD, the range of possible values of the additional convective flux, $u_i \psi(r) \frac{2\phi - \phi_{E}}{\phi_{P}}$, must be limited. Therefore, $\psi(r)$ is called a flux limiter function. It was noted by Sweby (1984) that for a scheme to be second order accurate, it has to pass through the point $(1,1)$ in the $r - \psi$ diagram, Fig. 4.3, which also show the region of second order accurate TVD schemes.

4.3.4c. Temporal discretization. The temporal domain is similarly to the spatial domain discretized into finite time steps. Since the value of $\phi$ at time $t^n$ (where the superscript $n$ denotes the current time step) only can depend on previous time steps, temporal discretization is always extrapolation. In order to simplify the process Eqn. (4.26) is reformulated as:

$$\int_V \frac{\partial \phi}{\partial t} dV = F(\phi) \quad (4.34)$$
where $F$ is a function of $\phi$ (advective and diffusive fluxes). The time derivative is assumed constant over the control volume:

$$\int_V \frac{\partial \phi}{\partial t} dV \approx \frac{\partial \phi}{\partial t} \delta V$$

which leads to:

$$\frac{\partial \phi}{\partial t} \approx \frac{1}{\delta V} [F(\phi)]$$

Temporal schemes are divided into explicit and implicit schemes. An explicit scheme estimate the values of $\phi$ at the next time step only using the values of $\phi$ at the current and previous time steps, i.e.:

$$\phi^{n+1} = F(\phi^n, \phi^{n-1}, ...)$$ (4.37)

Implicit schemes also use the values at the next time step:

$$\phi^{n+1} = F(\phi^{n+1}, \phi^n, \phi^{n-1}, ...)$$ (4.38)

Consequently, an iterative approach is necessary for implicit time schemes, while the results can be solved directly using an explicit scheme. However, in order to avoid divergence the time step for explicit schemes is limited by the Courant-Friedrich-Lewis limit (CFL), which for compressible flows reads:

$$CFL = \frac{(|U| + a)\Delta t}{\Delta x} < 1$$ (4.39)

where $U$ is the convective velocity, $a$ is the speed of sound, $\Delta x$ is the cell size and $\Delta t$ is the time step. Physically, this means that information is not allowed to travel longer than the cell length ($\Delta x$) during the time $\Delta t$. However, for LES a convective Courant number below 1 is recommended to resolve the smallest eddies.
A simple implicit temporal scheme is the implicit Euler scheme:

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = F(\phi^{n+1})$$  \hspace{1cm} (4.40)

4.3.5. Standard computational setup

In the work presented in this thesis the SGS stresses were modeled implicitly, and the PISO algorithm was used to couple pressure and velocity, Sec. 4.3.2c and 4.3.3, respectively. Convective fluxes were calculated using the TVD scheme with the limiter function introduced by Sweby (1984) with $\beta = 1$. The Euler implicit scheme was used for temporal discretization. The low order can be motivated by the very small time step from the CFL limit. Generally, at small time steps the numerical diffusion from the spatial scheme is of greater importance as compared to the temporal scheme. Additionally, the PISO algorithm, Issa (1986), increases the temporal accuracy one order for each extra outer iteration (corrector step).
In order to obtain extract relevant information, different methods have been used to post-process the results.

The coordinate system shown in Fig. 5.1 is used throughout this thesis.

5.1. Vorticity

Studying the vorticity and vorticity equation during compression can provide additional understanding of the flow physics during compression. Since turbulence cannot exist without vorticity, the change of vorticity is closely related to the change in turbulence and mixing. Or as defined by Bradshaw (1972): “Turbulence is a three-dimensional time-dependent motion in which vortex stretching causes velocity fluctuations to spread to all wavelengths between a minimum determined by the viscous forces and a maximum determined by the boundary conditions of the flow. It is the usual state of fluid motion except at low Reynolds numbers.”

Vorticity is defined as the curl of the velocity:

$$\Omega = \nabla \times \vec{u}$$  \hspace{1cm} (5.1)
5. EVALUATION METHODS

The vorticity equation is obtained by taking the curl of the momentum equation, Eqn. (4.2), written as:

\[
\frac{\partial \vec{\omega}}{\partial t} + (\vec{u} \cdot \nabla) \vec{\omega} = -\frac{1}{\rho} \nabla p + \nabla \times \left( \frac{\nabla \cdot \vec{\tau}}{\rho} \right)
\]  

(5.2)

The most important terms during compression are vortex stretching and vorticity-dilatation. Using the continuity equation, Eqn. (4.1), the vorticity-dilatation can be rewritten as:

\[
- \vec{\omega} (\nabla \cdot \vec{u}) = \vec{\omega} \frac{1}{\rho} \frac{D\rho}{Dt} = - \vec{\omega} \frac{1}{V} \frac{DV}{Dt}
\]

(5.3)

where \( V \) is the volume of a control volume, i.e. cell volume for CFD simulations. If the in-cylinder flow speed is much smaller than the speed of sound, which is the case for heavy-duty engines, the change in cell volume and density is equal the change in total cylinder volume and mean density.

5.2. Kinetic energy

The flow inside an engine can be assumed to consist of three different structures. Two large-scale motions, a rotating motion and a compressing motion caused by the piston. Turbulence (the third structure) and other unstructured flow motions are assumed to be the remainder of the flow. Hence, all kinetic energy is assumed to be contained in these three flow structures. The SGS turbulent kinetic energy is neglected, since most of the spectrum is resolved, see Sec. 7.2. The proposed methodology is validated in Sec. 7.3. The total kinetic energy is calculated as:

\[
E = \frac{1}{2} \sum_{k=1}^{N} \vec{u}_k \cdot \vec{u}_k \rho_k \Delta V_k
\]

(5.4)

where subscript \( k \) is the cell index, \( N \) the total number of cells, \( \vec{u}_k \) the velocity vector, \( \rho_k \) the density and \( \Delta V_k \) the cell volume. The kinetic energy of the rotational structure is formulated as:

\[
E_{\Omega} = \frac{1}{2} \vec{\Omega}^T \mathbf{I} \vec{\Omega}
\]

(5.5)

where \( \mathbf{I} \) is the moment of inertia around the center of gravity of the gas mixture within the cylinder:

\[
\mathbf{I} = \sum_{k=1}^{N} \left( \vec{r}_k \cdot \vec{r}_k \mathbf{E} - \vec{r}_k \vec{r}_k^T \right) \rho_k \Delta V_k
\]

(5.6)
where $\vec{r}$ is the distance from the center of gravity and $E$ is the identity tensor. Assuming solid body type rotation, (mean) angular velocity around the center of gravity, $\vec{\Omega}$, is calculated as:

$$\vec{\Omega} = \mathbf{1}^{-1}\vec{L}$$

(5.7)

where $\vec{L}$ is the angular momentum around the center of gravity:

$$\vec{L} = \sum_{k=1}^{N} \vec{r}_k \times \vec{u}_k \rho_k \Delta V_k$$

(5.8)

Kinetic energy around the tumble axis, $E_{\Omega,t}$, is formulated as:

$$E_{\Omega,t} = \frac{1}{2}(\Omega_x L_x + \Omega_y L_y)$$

(5.9)

where the $\Omega_x$ and $\Omega_y$ is the (mean) angular velocity perpendicular to the cylinder axis, i.e. $x$ and $y$-axis.

Kinetic energy introduced by the piston through compression is caused by the velocity of the flow introduced by the piston motion. Close to the piston wall the velocity of the gas is equal to that of the piston. Next to the cylinder head the velocity of the gas has to be zero. Between cylinder head and piston a linear velocity profile can be assumed due to the low velocity of the piston compared to the speed of sound. The kinetic energy of the piston induced motion can thus be calculated as:

$$E_{\text{pist}} = \frac{1}{2} \int_0^{z_p} (U_p z) \rho Adz = \frac{1}{2} U_p^2 m$$

(5.10)

where $U_p$ is the piston speed, $z$ is the distance from the cylinder head, $z_p$ is the distance between cylinder head and piston, $A$ is the cylinder area and $m$ is the total in-cylinder fluid mass.

The kinetic energy of the turbulence can then be calculated as:

$$e = E - (E_{\Omega} + E_{\text{pist}})$$

(5.11)

This approximation is correct if no other coherent motions exist and if the rotational motion is of solid body type. The error in kinetic energy of the rotational motion is less then 2% if the angular velocity varies with radius as $\Omega = A \cdot r^b$, where $A$ is a constant and $-0.5 < b < 0.65$. This is a good approximation for the studied flow (neither squish nor fuel spray), see Sec. 7.3 for validation.

In order to normalize the results an energy based on the mean piston speed ($MPS$) and in-cylinder mass ($m$) is formulated as:

$$\text{MPEnergy} = \frac{1}{2} m MPS^2$$

(5.12)

where $MPS$ is the mean piston speed, calculated as:

$$MPS = 2 \cdot S \cdot N/60$$

(5.13)
where $S$ is the engine stroke and $N$ is the engine speed.

5.3. The swirl number and swirl coefficient

Five different parameters have been used to characterize the flow entering the cylinder, a swirl coefficient ($C_s$), mean and instantaneous Thien swirl number ($SN_t$ and $SN_{tI}$), instantaneous swirl number ($SN_I$) and a swirl number ($SN$), all of which will be defined in the following subsections.

5.3.1. Swirl number

The in-cylinder swirl number is defined as the ratio between the angular velocity of the flow around the cylinder axis, $\Omega_{Swirl}$, and the angular velocity of the crank shaft, $\Omega_E = 2\pi \cdot N/60$, Eqn. (3.1). The characteristic angular velocity of the flow ($\Omega$) can be calculated as:

$$\vec{\Omega} = I^{-1}\vec{L}$$

where $I$ is the moment of inertia, $\vec{L}$ is the angular momentum around the center of gravity, evaluated as:

$$\vec{L} = \iiint_V \vec{r} \times \vec{u} \rho dV$$

where $\vec{u}$ is the fluid velocity, $\vec{r}$ is the distance from the center of gravity, $\rho$ is the density and $V$ is the cylinder volume. The moment of inertia, $I$, is calculated as:

$$I = \iiint_V (\vec{r} \cdot \vec{r}) - (\vec{r} \cdot \vec{r})^T \rho dV$$

where $\mathbf{E}$ is the identity tensor. The moment of inertia of a gas mixture within a cylinder around it main axis $(z)$ is:

$$I_{zz} = \frac{1}{2} m R^2$$

where $m$ is the mass of the gas mixture within the cylinder (air) and $R$ is the cylinder radius. Angular velocity around the swirl (cylinder) axis is calculated as:

$$\Omega_{Swirl} = \vec{\Omega} \cdot \vec{e}_z$$

where $\vec{e}_z$ is the unit vector along the cylinder axis. Angular velocity around the cylinder axis can also be calculated as:

$$\Omega_{Swirl} = L_z/I_{zz}$$
5.3. THE SWIRL NUMBER AND SWIRL COEFFICIENT

5.3.2. Measuring the swirl coefficient in a test rig

In order to calculate the swirl number, Eqn. (3.1), the flow field within the entire cylinder needs to be known. Due to practical problems, volume measurements within a real engine cylinder are extremely difficult. Therefore, the swirl number, $SN$, is approximated from the flow measured in a steady test rig. However, using traditional measuring techniques it is not possible to measure a crank angle resolved flow of angular momentum with moving valves and changing mass flow. Therefore, the angular momentum flow is measured for steady mass flow and normalized to a swirl coefficient, $Cs$, at fixed valve openings. The angular momentum flow into the cylinder (for steady state conditions) is calculated (and can be measured using a honeycomb torque meter) as:

$$M_{z,d}(\Lambda) = \int_A (u_y \cdot r_x - u_x \cdot r_y)(-\rho u_z) dA$$

(5.19)

where the subscript $d$ indicates measured/calculated in steady condition, the subscript $z$ indicates the cylinder axis, $A$ is the cylinder area and $\Lambda$ is the valve lift. Assuming solid body rotation and constant axial velocity:

$$M_{z,d}(\Lambda) = \Omega_d(\Lambda) \int_0^R \int_0^{2\pi} \rho(-u_z)r^3drd\phi = \Omega_d(\Lambda)Q_d(\Lambda) \frac{R^2}{2}$$

(5.20)

where $R$ is the cylinder radius. The forementioned equation can be rewritten to get the steady angular velocity, $\Omega_d$, as:

$$\Omega_d(\Lambda) = \frac{M_{z,d}(\Lambda)}{\frac{1}{2}Q_d(\Lambda)R^2}$$

(5.21)

where $Q_d$ is the measured mass flow into the cylinder. However, in order to obtain a swirl coefficient, $Cs$, the angular velocity of the flow should be normalized with an engine speed, $n$. The engine speed is set to correspond to the mean piston speed ($MPS$) and mass flow ($Q_{mp}$) of the engine.

$$MPS = 2S \cdot \frac{n}{60} \quad Q_{mp} = MPS \cdot \pi R^2 \rho$$

(5.22)

$$n = \frac{30Q_{mp}}{S\rho \pi R^2}$$

where $S$ is the engine stroke. By assuming that $Q_{mp} = Q_d$, the swirl coefficient can be written as:

$$Cs = \frac{\frac{\Omega_d}{2\pi n/60}}{\frac{\Omega_d \cdot S}{\pi MPS}} = \frac{2S \cdot \rho M_{z,d}}{Q_d^2}$$

(5.23)

5.3.3. Calculating Thien swirl number

Due the sinusoidal piston motion and the varying valve lift the swirl coefficients need to be weighted and rescaled to obtain a correct swirl number as in Eqn. (3.1). Following the procedure described by Thien (1965).

$$L_z = \int_{\alpha_{IVO}}^{\alpha_{IVC}} M_z(\alpha) \frac{dt}{d\alpha} d\alpha$$

(5.24)
where \( t \) is time, \( \alpha \) is the Crank Angle in radians, the subscript IVO is Inlet Valve Opening, IVC is Inlet Valve Closing. From Eqn. (5.20) we know that:

\[
M_z(\alpha) = \frac{\Omega(\alpha)Q(\alpha) \cdot R^2}{2} \tag{5.25}
\]

where \( \Omega(\alpha) \) is the angular velocity of the flow entering the cylinder at crank angle \( \alpha \). The angular velocity of the flow entering the cylinder can be assumed to scale with the piston speed, \( U_{pist} \), and can thus be written as:

\[
\Omega(\alpha) = \Omega_d(\Lambda) \frac{U_{pist}(\alpha)}{MPS} \tag{5.26}
\]

where \( \Omega_d(\Lambda) \) can be measured. Flow of angular momentum as a function of crank angle can be written as:

\[
M_z(\alpha) = QR^2 \frac{\Omega_d(\alpha)}{MPS} U_{pist}(\alpha) = \frac{\pi R^2 \rho R^2 \Omega_d(\alpha)}{MPS} U_{pist}^2(\alpha) \tag{5.27}
\]

where the piston velocity with respect to crank angle can be written:

\[
\frac{dz_{pist}}{d\alpha} = \frac{U_{pist}}{d\alpha} = \frac{U_{pist}}{30 \pi N} \tag{5.28}
\]

Therefore, a final expression for the total angular momentum entering during one stroke can be written as:

\[
L_z = \frac{\pi^2 R^2 \rho R^2}{2S} \int_{IVO}^{IVC} \Omega_d(\alpha) \left[ \frac{dz_{pist}(\alpha)}{d\alpha} \right]^2 d\alpha \tag{5.29}
\]

and the expression for the mean swirl number (Thien swirl, \( SN_t \)) becomes:

\[
SN_t = \frac{\pi}{S} \int_{IVO}^{IVC} C_s(\alpha) \left[ \frac{dz_{pist}(\alpha)}{d\alpha} \right]^2 d\alpha \tag{5.30}
\]

### 5.3.4. Instantaneous swirl number

As the swirl number for an entire stroke can be calculated as:

\[
SN = \frac{L_z}{\frac{1}{2} R^2 \cdot m \cdot 2\pi N/60} = \frac{\int_{IVO}^{IVC} M_z(\alpha) \frac{d\alpha}{d\alpha} d\alpha}{\frac{1}{2} R^2 \cdot \left[ \int_{IVO}^{IVC} Q(\alpha) \frac{d\alpha}{d\alpha} \right] \cdot 2\pi N/60} \tag{5.31}
\]

the instantaneous swirl number is introduced as:

\[
SN_t(\alpha) = \frac{M_z(\alpha)}{\frac{1}{2} \cdot R^2(Q(\alpha))2\pi \cdot N/60} \tag{5.32}
\]

Similarly, an instantaneous Thien swirl number is introduced as:

\[
SN_{tI}(\alpha) = \frac{\pi}{S} C_s(\alpha) \frac{dz_{pist}(\alpha)}{d\alpha} \tag{5.33}
\]
5.4. Turbulent anisotropy

According to Lumley & Newman (1977) the level of turbulence anisotropy can be quantified using the anisotropy tensor:

\[ b = \frac{\langle \vec{u}' \cdot \vec{u}' \rangle}{q^2} - \frac{1}{3} \mathbf{E} \]  \hspace{1cm} (5.34)

and its scalar invariants:

\[ II = \mathbf{b} : \mathbf{b} \]  \hspace{1cm} (5.35)
\[ III = (\mathbf{bb}) : \mathbf{b} \]  \hspace{1cm} (5.36)

where : indicates the double dot product, \( \vec{u}' \) is the fluctuating velocity vector, \( \mathbf{E} \) is the identity tensor and \( q^2 \) is the trace of the Reynolds stress tensor, \( q^2 = \langle \vec{u}' \cdot \vec{u}' \rangle \). It can be concluded that \( \mathbf{b} \) has zero trace and vanishes, together with \( II \) and \( III \), for isotropic turbulence. Additionally, each component in \( \mathbf{b} \) cannot be smaller than \( -1/3 \) or larger than \( 2/3 \). Hence, the possible physical values for \( II \) and \( III \) is bounded by axisymmetric Reynolds stress tensor and two-component turbulence. Where the axisymmetric boundary can be calculated as:

\[ II_{axi} = \frac{3}{2} \left( \frac{4}{3} |III| \right)^{2/3} \]  \hspace{1cm} (5.37)

and two component turbulence:

\[ II_2 = \frac{2}{9} + 2III \]  \hspace{1cm} (5.38)

If these boundaries are plotted one obtain what is now known as the Lumley triangle, Fig. 5.2.

\[ \text{Figure 5.2. Anisotropy-invariant map (Lumley triangle).} \]
5. EVALUATION METHODS

5.5. Turbulent length and time scales

In Paper 3 the integral length scale of the tangential fluctuations was used to characterize the length scale of the turbulence. First the autocorrelation function (matrix) is defined by:

\[ R(r, t) = \langle \vec{u}'(\vec{x}, t) \vec{u}'(\vec{x} + \vec{r}, t) \rangle \]  

(5.39)

where \( \vec{r} \) is the distance between two points in the flow. Longitudinal length scale is obtained if \( \vec{r} \) is parallel to \( \vec{u}' \) and transverse if \( \vec{r} \) is perpendicular to \( \vec{u}' \). Calculation of longitudinal integral length scale was conducted along circumferential lines in a plane in the center of the cylinder. Deviation from the mean tangential velocity (\( u'_\Theta \)) was used as turbulent fluctuation (\( \vec{u}' \)). The longitudinal integral length scale of the velocity can then be calculated as:

\[ \Lambda_{\Theta \Theta} = \int_{0}^{\infty} R_{\Theta \Theta}(r, t) / \langle u'^2_\Theta \rangle \, dr \]  

(5.40)

Assuming that the turbulence is frozen in the mean flow, a time scale, \( \tau_\Theta \), can be calculated as:

\[ \tau_\Theta = \Lambda_{\Theta \Theta} / \langle U_\Theta \rangle \]  

(5.41)
Flow under consideration

In this chapter the geometrical, numerical and experimental setups used are presented. The setup regarding the dynamic effects during intake (Paper 1, 4 and 6) are presented in Sec. 6.1. The effects of compression on large-scale flow structures and turbulence presented in Paper 3 and 5 are described in Sec. 6.2. The setup of a full cycle simulation which was used to validate the methodology for estimating turbulent kinetic energy is presented in Sec. 6.3.

6.1. Dynamic effects during the intake

Traditionally, the measurements of swirl in CI engines have been made in steady swirl test rigs, see Sec. 3.5.3. As the name suggests, the flow is statistically stationary and consequently dynamic effects are not accounted for. The aim of the calculations of the intake geometry was to study the dynamic effects that can be expected within an engine but not captured in a steady swirl rig, Sec. 1.1. Calculations of the swirl number have been performed using the methodology described in Sec. 5.3. Understanding the expected dynamic effects during intake is important in order to compare swirl numbers measured in a steady test rig and the swirl numbers used as initial conditions in sector models, see Sec. 3.5.4.

6.1.1. Geometry

LES computations and PIV measurements were carried out on a steady swirl flow rig. For the experiments an impulse torque swirl rig located at Scania was used, consisting of a cylinder head mounted on a transparent glass cylinder having a diameter of 130 mm and a height of 220 mm. The glass cylinder was placed on a stagnation chamber from which air was drawn by a pressure drop controlled fan, see Fig. 6.1. A similar model was used in the computations. However, in the simulations, the cylinder was extruded an additional 3 diameters and the stagnation chamber removed, as shown in Fig. 6.2.
6. FLOW UNDER CONSIDERATION

Figure 6.1. PIV measurement setup. Position and size of the measurement plane, with position of the valves marked. Flow inlet direction indicated by arrows.

Figure 6.2. Geometry used in the simulations.
6.1. DYNAMIC EFFECTS DURING THE INTAKE

6.1.2. PIV setup

A stereo PIV setup was used, having the two cameras mounted in a backward-backward configuration as shown in Fig. 6.1. The double pulse ND:Yag Laser created a 2 mm thick laser sheet located 165 mm downstream (below) the cylinder head and 5 µm atomized alcohol droplets were used to seed the flow. As displayed in Fig. 6.1 there is no inlet channel in which seeding droplets can be introduced. To overcome this, the entire room was filled with smoke during the experiments. Due to reflections of the laser on the glass cylinder black tape was used to stop reflections to hit the camera. As a consequence, not all of the cylinder could be measured, see Fig. 6.1. A thorough description of the PIV measurement setup can be found in Vernet (2012).

6.1.3. Boundary conditions and LES setup

Several boundary conditions were used in order to study the dynamic effects during intake. First, LES calculations and PIV measurements were used to characterize swirl fluctuations for different valve lifts in a statistically stationary flow, Sec. 6.1.3a. Second, the effect of flow acceleration over fixed valves were studied using LES with time varying boundary conditions, Sec. 6.1.3b. Finally, the dynamic effects of valve motion were studied for otherwise stationary conditions, Sec. 6.1.3c. A maximum convective Courant number below 0.7 were used at all times, in turn leading to a significantly lower convective Courant number for most of the geometry.

6.1.3a. Statistically stationary flow, LES and PIV. The cases presented in Table 6.1 were measured using PIV with a resolution of $0.6 \times 0.6 \text{ mm}^2$. The three chosen valve lifts correspond to an area opened by the valves (curtain area) to intake port area ratio of 50, 100 and 150 %. Large eddy simulations were carried out on cases $A_2$, $B_3$ and $C_3$ in Table 6.1. Mass flow and temperature were set at the inlet, at the outlet a static pressure matching the experiments was applied.

6.1.3b. Effects of flow acceleration on swirl. A fixed valve lift of 15 mm was used to study the effect of flow acceleration on angular momentum around the cylinder axis (swirl). At the inlet a non-reflective static pressure of 355 kPa was set. Two cases with different velocity outlet boundary condition were simulated. First, a steady velocity equal to the mean piston speed of an engine defined by the parameters in Table 6.2, was applied. Second, the effects of an outlet velocity corresponding to the piston speed of several consecutive intake cycles of the same engine were investigated. The outlet velocity was smoothed between the cycles in order to avoid discontinuities in the acceleration. Each cycle (intake stroke) was set to start at 0 CAD (TDC) and finish at BDC (180 CAD). The prescribed outlet velocity boundary condition had the form
Table 6.1. Experimental setup

<table>
<thead>
<tr>
<th>Case</th>
<th>Valve lift [mm]</th>
<th>Pressure drop [Pa]</th>
<th>Air density [kg/m³]</th>
<th>Room temperature [°C]</th>
<th>Bulk velocity [m/s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>A₁</td>
<td>5</td>
<td>800</td>
<td>1.2</td>
<td>21</td>
<td>2.61</td>
</tr>
<tr>
<td>A₂</td>
<td>5</td>
<td>1400</td>
<td>1.2</td>
<td>21</td>
<td>3.43</td>
</tr>
<tr>
<td>A₃</td>
<td>5</td>
<td>2400</td>
<td>1.2</td>
<td>21</td>
<td>4.57</td>
</tr>
<tr>
<td>B₁</td>
<td>11</td>
<td>800</td>
<td>1.2</td>
<td>21</td>
<td>4.02</td>
</tr>
<tr>
<td>B₂</td>
<td>11</td>
<td>1400</td>
<td>1.2</td>
<td>21</td>
<td>5.36</td>
</tr>
<tr>
<td>B₃</td>
<td>11</td>
<td>2400</td>
<td>1.2</td>
<td>21</td>
<td>7.19</td>
</tr>
<tr>
<td>C₁</td>
<td>15</td>
<td>800</td>
<td>1.2</td>
<td>21</td>
<td>4.26</td>
</tr>
<tr>
<td>C₂</td>
<td>15</td>
<td>1400</td>
<td>1.2</td>
<td>21</td>
<td>5.67</td>
</tr>
<tr>
<td>C₃</td>
<td>15</td>
<td>2400</td>
<td>1.2</td>
<td>21</td>
<td>7.59</td>
</tr>
</tbody>
</table>

Figure 6.3. Outlet boundary velocity for the steady (dashed) and original varying case (solid).

as presented in Fig. 6.3. 24 cycles were simulated for the original case, out of which the first four cycles were discarded to remove initialization effects.

6.1.3c. Dynamic effects of flow valve motion. A total to static pressure drop of 1500 Pa over the geometry was set as boundary condition, and the simulations were run incompressible. Inlet valves were moving in a sinusoidal motion between 5 and 15 mm valve lift. Each cycle was of similar length as in Sec. 6.1.3b leading to valve lifts at 0, 90 and 180 CAD that were 5, 15 and 5 mm, respectively (Fig. 6.4).
6.1. DYNAMIC EFFECTS DURING THE INTAKE

Table 6.2. Engine parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore, $D$ [mm]</td>
<td>130</td>
</tr>
<tr>
<td>Stroke, $S$ [mm]</td>
<td>160</td>
</tr>
<tr>
<td>Conrod length, $L$ [mm]</td>
<td>255</td>
</tr>
<tr>
<td>Engine speed, $N$ [rpm]</td>
<td>1200</td>
</tr>
<tr>
<td>Port fluid temperature [K]</td>
<td>318.15</td>
</tr>
<tr>
<td>Compression ratio [-]</td>
<td>17.8</td>
</tr>
<tr>
<td>$Re = \frac{(\rho MPS \cdot D)}{\mu}$ [-]</td>
<td>55000</td>
</tr>
</tbody>
</table>

Figure 6.4. Valve lift curve.
6.2. Effects of compression

The effects of compression on a tumbling flow at low compression ratios have been studied extensively as mentioned in Sec. 3.2. However, studies of flow fields more relevant to heavy-duty diesel engines, i.e. tilted rotation, have mainly been performed on complex geometries where less control of the flow structures have been possible, see Sec. 3.5.5. This section describes the setup of the first systematic study of the effect of compression of tilted rotational motions. Additionally, the effect of compression on isotropic turbulence in swirling motions of different strength were investigated.

6.2.1. Geometry

In order to study the effects of compression alone without adding the extra complexity of squish, the used geometry was a simple cylinder. Cylinder specifications as well as running conditions can be found in Table 6.2. Crank angles 180-380 were simulated, where 180° and 360° correspond to BDC and TDC, respectively. Thus, 20 crank angles were simulated after firing TDC.

6.2.2. Numerical setup

An Arbitrary Lagrangian-Eulerian (ALE) mesh motion technique were used, i.e. the grid is compressed with geometry change. In order to avoid cells with too high aspect ratio, which would reduce accuracy significantly, see Celik et al. (2005), the results were mapped to a new grid once the geometry had been compressed by a factor two. This lead to the result being remapped to a new grid four times, i.e. five different grids were used. Adiabatic no-slip boundary condition was used for all walls. The boundary layer was not resolved as this would require a significant amount of cells and the effect were deemed small, see Lumley (1999). A convective Courant number below 0.2 were used at all times.

6.2.3. Grid

Grids were created by setting an apparent cell size, $\Delta x$, the number of cells (#) for the grid at TDC in direction A-C was calculated as:

\[
\#_A : \frac{2\pi R_0}{6\Delta x} \tag{6.1}
\]

\[
\#_B := \frac{2\pi R_0}{9\Delta x} \tag{6.2}
\]

\[
\#_C : \sqrt{2} \frac{\Delta h}{\Delta x} \tag{6.3}
\]

where $\Delta h$ is the distance between piston and cylinder head at TDC (clearance). A graphical view of a 3 mm grid is depicted in Fig. 6.5.
6.2. EFFECTS OF COMPRESSION

6.2.4. Initial conditions

In order to study the effect of compression on different flow structures systematically, different initial conditions were introduced at BDC and studied during compression. First, the effects of compression on turbulence in a swirling flow were studied, Sec. 6.2.4a. Second, the effect of tilt angle on flow homogeneity, turbulence and mixing for a rotating flow undergoing compression, Sec. 6.2.4b.

6.2.4a. Isotropic turbulence in a swirling flow. Solid body type rotation with swirl numbers, Eqn. 3.1, corresponding to 0, 1 and 4 were initialized. Isotropic turbulence with turbulent kinetic energy per unit mass of 23 m$^2$/s$^2$ was superimposed on the swirling motion. In order to calculate turbulent quantities 1, 6 and 3 compressions with slightly varying initial turbulence were performed for the three cases, respectively. Mean in-cylinder temperature and pressure were 300 K and 10$^5$ Pa, respectively.

6.2.4b. Tilted rotating motion and effect of rotational strength. Six different tilt angles, 0°, 9°, 19°, 34°, 61° and 90°, were simulated. The 0° tilt angle case was given a swirl number of two. The other cases were made to contain the same amount of total kinetic energy but different tilt angle as this case.

---

1Söder et al. (2013b), figure 5 with kind permission from ASME
This lead to initial swirl numbers of 2, 1.9, 1.7, 1.3, 0.65 and 0, respectively. Corresponding tumble numbers were 0, 0.3, 0.6, 0.9, 1.18 and 1.23, respectively.

In order to examine the effect of initial kinetic energy, two more sets of calculations were performed. In these sets (which included tilt angles 9°, 19°, 34°, 61° and 90°) the kinetic energy was reduced to 25 and 56 % of the kinetic energy of the original cases, respectively. This was done by multiplying the velocity field of the original cases with 0.5 and 0.75, respectively.

The effect of tumble strength was studied by scaling (multiplying) the velocity field of the 90° tilt case with 0.25, 0.5, 0.75, 0.96, 1 and 1.5 and used as initial conditions. This correspond to tumble numbers of 0.3, 0.6, 0.9, 1.18, 1.23 and 1.85.

Additionally, the velocity for the 9°, 19°, 34° and 61° cases were also scaled with 0.5 and 0.75. This scaling was made in order to study the effect of compression on rotational tilts of different strengths.

The different combinations of swirl, tumble and tilt are illustrated in Fig. 6.6

![Figure 6.6. Computational cases, color indicating tilt angle.](image)

In order to study mixing, two passive scalars were initialized at BDC. The first, $C_1$, was initialized in the outer region of the cylinder and the second, $C_2$, in the regions with lowest velocity. Streamlines of the initial velocity field as well as passive scalar $C_2$ is visualized in Fig. 6.7. Both scalars have an initial value of either 0 or 1 with a mass averaged value of 0.5 in the cylinder. Mean in-cylinder temperature and pressure were 300 K and $10^5$ Pa, respectively.
6.2. EFFECTS OF COMPRESSION

Figure 6.7. Starting conditions, Streamlines and position of passive scalar $C_2$. From top left, 0°, 9°, 19°, 34°, 61°, 90°.²

²Söder et al. (2014), figure 1 with kind permission from SAE International
6. FLOW UNDER CONSIDERATION

6.3. Full cycle simulation

In order to validate the methodology of calculating turbulent kinetic energy presented in Sec. 5.2, the proposed methodology was compared to turbulence calculated using a phase-averaged flow, see Sec. 3.4. Several full cycle strokes were necessary to calculate a phase-average and subsequent turbulence.

6.3.1. Geometry

The intake geometry was the same as presented in Sec. 6.1. However, instead of an extruded cylinder a flat piston was used to drive the flow, Fig. 6.8. The intake valves were fixed at 15 mm valve lift and closed instantaneously at BDC, the geometry was such that the lowest part of the valves was at the same height as the cylinder head. During compression the geometry was reduced to a cylinder similar as presented in Sec. 6.2. The engine was running using parameters presented in Table 6.2.

![Figure 6.8. Geometry at crank angle 100.](image)

6.3.2. Numerical setup and boundary conditions

The grids during intake were created using an automatic mesher with a cell size of 0.6 mm, consisting of 7.6 million and 18.3 million cells at TDC and BDC, respectively. During compression the same grids as presented in Sec. 6.2 were used. The expansion and exhaust strokes were calculated using very course grids. The course grids were used since focus was not on these parts of the engine cycle nor was combustion simulated, why the exact values are of lesser importance. An Arbitrary Lagrangian-Eulerian (ALE) mesh motion technique was used. Five different meshes were used for both intake and compression. A total of 17 cycles were calculated where the first two cycles were discarded. Adiabatic no-slip boundary condition was used for all walls, the boundary layer was not resolved as this would require a significant amount of cells and...
the effect was deemed small, see Lumley (1999). At the inlet a non-reflective static pressure of 355 kPa was set. A convective Courant number below 0.7 were used at all times. Additionally, 5 cycles were run at 1800 rpm in order to study the effect of engine speed.
CHAPTER 7

Validation

In this chapter, the grid studies for the different cases investigated in this work is presented. The validation of the results obtained for the intake is presented in Sec.7.1. This is followed by the validation of the computational setup used to study the effects of compression, Sec. 7.2. Finally, the methodology for calculating turbulent kinetic energy is validated in Sec. 7.3.

7.1. Grid study, intake

To ensure that the energy containing eddies were resolved a mesh study was performed. The mesh was refined until the energy spectrum exhibited a resolved inertial subrange of a length over one decade as shown in Fig. 7.1. The results suggest that the 0.9 mm mesh is sufficient for resolving the energy containing scales of the flow field. To add margin the 0.7 mm mesh was further investigated, this mesh size was found to agree reasonable well with steady swirl test rig results, Fig. 7.2. This indicates that the 0.7 mm mesh was sufficiently fine to resolve the main features of the flow was thus chosen. For the moving valve case, a mesh size of 0.9 mm was used due to practical considerations.

Figure 7.1. Energy spectrum of the axial velocity in a point between the lower part of the valves (a) and in the center of the cylinder 0.5 diameters below cylinder head (b) for different mesh sizes, 11 mm valve lift (Söder et al. 2010).

54
Figure 7.2. Measured and calculated (0.7 mm grid) normalized mean swirl coefficient (Söder et al. 2010).
7. VALIDATION

7.2. Grid study, compression

Accurate LES simulations require that most of the turbulent kinetic energy is resolved. The methodology described in Sec. 5.2 was used to obtain an estimate of turbulent kinetic energy. Thereafter, a grid study was conducted in order to find a sufficiently fine grid, Fig. 7.3. Based on these results the 0.46 mm mesh was chosen as the difference between the 0.46 mm and 0.41 mm meshes was negligible. For the case of isotropic turbulence in a swirling flow it was possible to compare turbulent intensity at a specific radius during compression and turbulent intensity for different radii at TDC, Fig. 7.4

![Figure 7.3](image1.png)

**Figure 7.3.** Average cell size effect (during compression) on the kinetic energy of the different flow structures defined in Sec. 5.2 (Söder et al. 2013b).

![Figure 7.4](image2.png)

**Figure 7.4.** Turbulent intensity ($I_{u'}$), $\sqrt{\frac{1}{3}(u'^2 + u'_y^2 + u'_z^2)/MPS}$, at radius $R/R_0 = 0.3$ as a function of CAD (a) and at 360 CAD as a function of $R/R_0$ (b). 0.46 mm (-) and 0.41 mm (- -) grids (Söder et al. 2013a).
7.3. Validation of methodology

In order to validate the methodology for calculating the turbulent kinetic energy ($e$) described in Sec. 5.2, this method was compared to in-cylinder turbulence energy ($tke$) calculated using a phase-average as described in Sec. 3.4. In order to compare the results, $tke$ in all cells were summed over all $N$ cells inside the cylinder to form $TKE$, Eqn. (7.1).

$$TKE = \sum_{k}^{N} m_k tke_k$$

(7.1)

where $m_k$ is the mass inside cell $k$. The two methodologies are compared in Fig. 7.5 and it is shown that the proposed methodology in Sec. 5.2 is not valid during the intake stroke. However, at the end of compression, a perfect agreement is observed. Overall, reasonable agreement is obtained during the compression stroke.

![Figure 7.5. Comparison between calculating turbulent kinetic energy using the methodology described in Sec. 5.2 (dashed black, $e$) and from a phase-average, Sec. 3.4 (red, $TKE$), respectively. Estimated error in the mean is shown with lighter color.](image-url)
CHAPTER 8

Results and discussion

Results regarding what affects flow structures during the intake stroke is presented and discussed in Sec. 8.1. The subsequent influence of compression is presented in Sec. 8.2. Finally, the results from the full cycle simulation are presented in Sec. 8.3.

8.1. Results, Dynamic effects during intake

8.1.0a. Steady flow and fixed valves. Depending on valve lift, the flow into the cylinder exhibits clear differences, see Fig. 8.1. It can be seen that for the high valve lifts (11 and 15 mm) the flow enters the cylinder in the direction of the ports. However, for the 5 mm valve lift case the direction of the ports is not noticeable, the small curtain area (valve lift) has changed the direction of the flow completely. This leads to a drastic reduction in the swirl coefficient, Fig. 8.2.

For the 5 mm valve lift case the fluctuation in swirl coefficient is greater than the mean swirl coefficient. This indicates that the fluctuations in swirl coefficient is of greater importance than the mean value at low valve lifts.

Turbulent anisotropy is found to be significantly higher for the 5 mm valve lift case. For the higher valve lift cases turbulent fluctuations are mainly axisymmetric.

From the PIV measurements it is found that the driving pressure has a negligible effect on the (scaled) mean velocity, Fig. 8.3, as well as on turbulent anisotropy, Fig. 8.4. Therefore, the flow is Reynolds number independent meaning that a change in mass flow will have a negligible effect on the created flow structures, i.e. the swirl coefficient can be expected to be independent of engine speed. However, this is only true for a steady flow, which is the case in the test geometry, for the real dynamic case more effects can be observed. These effects are the motivation of the next sections.
8.1. RESULTS, DYNAMIC EFFECTS DURING INTAKE

Figure 8.1. Axial velocity and in-plane velocity vectors scaled with bulk velocity just inside the engine cylinder seen from above. From top left (a, b, c), 5, 11 and 15 mm valve lift, respectively (LES).

Figure 8.2. Instantaneous swirl coefficient from PIV of the reduced surface, Fig. 6.1. Rows correspond to 5, 11 and 15 mm valve lift, respectively.
Figure 8.3. Mean axial velocity scaled with bulk velocity, with in-plane velocity vectors. Rows correspond to pressure drops of 800, 1400 and 2400 Pa, respectively. Columns represent 5, 11 and 15 mm valve lift, respectively (Söder et al. 2010).
Table 8.4. Turbulent anisotropy invariant map, colors indicate the distance from the cylinder center in cylinder radii. PIV data, offset of 0.025 on the $x$-axis between presented cases. Rows correspond to 5, 11 and 15 mm valve lift, respectively.
8. RESULTS AND DISCUSSION

8.1.1. Effect of flow acceleration

8.1.1a. Overview. In Fig. 8.5 the phase-averaged velocity in plane A, Fig. 8.6, is plotted at different crank angles during the intake cycle. In the beginning of the cycle a small backflow can be seen. When the flow starts to flow into the cylinder it is mainly directed straight down, as the flow starts to decelerate it can be seen that the jet is more deflected towards on side.

Figure 8.5. Mean and phase-averaged velocity for the varying outflow case, plane A, Fig 8.6. From a to f, mean, at 0, 36, 72, 108 and 144 CAD. Please, notice the legend.
8.1. RESULTS, DYNAMIC EFFECTS DURING INTAKE

8.1.1b. Quantitative results. Phase-averaged results in plane B are plotted in Figs. 8.7-8.8. Results from the steady and time varying case have been plotted using a solid and dashed line, respectively. Unless otherwise specified the results from the time varying case are discussed and compared to the steady case.

In Fig. 8.7 (a) it can be seen that there is a phase lag between the outlet velocity and the mass flow passing through plane B. An overshoot and subsequent backflow can also be observed as a consequence of this. A phase lag can also be seen when comparing the mass flow and the total pressure loss (between inlet and plane B), Fig. 8.7 (b). It can also be noted that the total pressure drop is of the same order of magnitude as used in the experiments, Table 6.1. In Fig. 8.8 the flow of angular momentum (a) and swirl coefficient (b) are shown. The flow of angular momentum was found to be zero until 50 CAD when it starts to increase rapidly. This is caused by the backflow experienced until 30 CAD, thereafter it takes another 20 CAD for the cumulative volume flow to be greater than the volume above the valves, Fig. 8.9. Since the volume above the valves is unaffected by the port direction no angular momentum will be formed. The effect of this volume is not possible to notice in neither steady state simulations/experiments nor in full cycle simulations (or at least very difficult). If the measured swirl number is to be used in/or compared to CFD simulations the aforementioned effect is of great importance as the results may otherwise be misleading. In order to take this effect into account the ratio between this volume and total cylinder volume is proposed, and denoted port delay ratio, $R_p$:

$$ R_p = \frac{V_{ud}}{V_c} $$  \hspace{1cm} (8.1)
where $V_{ud}$ is the volume directly above the valves henceforth mentioned as undirected port volume, see Fig. 8.9, and $V_c$ is the cylinder volume at BDC.

The swirl coefficient was found to be lower during the acceleration phase as compared to the deceleration phase, see Fig. 8.8 (b). The difference in swirl coefficient is caused by the effect of the pressure gradient necessary to accelerate the flow. A favorable pressure gradient directs the flow downwards into the cylinder increasing axial momentum flow while a unfavorable pressure gradient reduces axial momentum flow and thus increasing the ratio of angular momentum flow to axial momentum flow.

Figure 8.7. Phase-averaged mass flow (a) and total pressure drop (b). Black solid and dashed lines correspond to steady and varying case, respectively. Blue dashed line corresponds to outlet mass flow. Purple and red dashed lines show the crank angle when the undirected port volume has been evacuated and at the start of outlet deceleration, respectively. Results for the different cycles shown using a slightly lighter color.
8.1. RESULTS, DYNAMIC EFFECTS DURING INTAKE

Figure 8.8. Flow of angular momentum (a) and swirl coefficient, $C_s$, (b). Black solid and dashed lines correspond to steady and varying case, respectively. Blue dashed line corresponds to outlet momentum flow estimated using Eqn. 5.27. Purple and red dashed lines show the crank angle when the undirected port volume has been evacuated and at the start of outlet deceleration, respectively. Results for the different cycles shown using a slightly lighter color.

Figure 8.9. Undirected port volume, $V_{ud}$. 
8. RESULTS AND DISCUSSION

8.1.2. Effect of valve motion

The phase lag between pressure and mass flow also has an effect when studying the effect of valve motion. It can be seen that the mass flow is significantly higher during valve closing as compared to the same valve lift during valve opening, Fig. 8.10a. Also, the swirl coefficient is found to be significantly higher during valve closing as compared to valve opening, Fig. 8.10b. It can be seen that during valve opening the recirculation zone under the valve seats are larger as compared to valve closing, Fig. 8.11. If the flow had not been found to be Reynolds number independent, Sec. 8.1.0a, this would be attributed to the change in mass flow. Instead the different swirl coefficients should be attributed to hysteresis or to the adverse pressure gradient ($\frac{dp}{dz} < 0$) inside the cylinder during valve closing, see Fig. 8.11. An adverse pressure gradient leads to a deceleration of the axial velocity and thus are increasing the swirl coefficient.

**Figure 8.10.** Phase-averaged volume flow (a) and phase-averaged swirl coefficient (b), individual cycles in gray. Vertical lines at crank angles 45°, 90° and 135°.
Figure 8.11. Phase-averaged velocity with streamlines (left) and pressure (right) at 45, 90 and 135 CAD.
8. RESULTS AND DISCUSSION

8.2. Results, Effects of compression

The effects of compression on isotropic turbulence in a swirling flow are presented first, see Sec. 8.2.1. In this section, the effect of compression on isotropic turbulence in a pure swirling with a swirl number of one is presented first. Thereafter, the effect of changing swirl number is presented. In Sec. 8.2.2 the effect of tilted rotation undergoing compression is presented. The effect of rotational strength is presented last in Sec. 8.2.3.

8.2.1. Isotropic turbulence in a swirling flow

8.2.1a. Isotropic turbulence in a swirling flow with a swirl number of one. The evolution of the flow velocity magnitude during compression can be seen in Fig. 8.12. It is observed that the initial high turbulence levels dissipate quickly and the swirl is becoming visible. The dissipation can also be seen in Fig. 8.13, where the turbulent fluctuations are presented. However, around 280 CAD the fluctuations in the axial velocity component starts to increase, this is followed by an increase in fluctuations in the other velocity components, see Fig. 8.13. In Fig. 8.14 it can be seen that the longitudinal length scale of the tangential fluctuations is increasing until 320 CAD when the length scale start to decrease.

Figure 8.12. Velocity magnitude in the cylinder mid-plane at different crank angles. Crank angles, from left to right starting at top-left, 180, 190, 220, 280, 330, 350, 360, 380. SN = 1 (Söder et al. 2013a).
8.2. RESULTS, EFFECTS OF COMPRESSION

8.2.1b. Effect of swirl number. An increase in swirl number is found to increase dissipation of the turbulent fluctuations, Fig. 8.15. Contrary to the results of the low and zero swirl cases the strongest fluctuations are found in the tangential velocity component, Fig. 8.16. Why the other components start to increase is not completely understood, but it may be caused by the formation of secondary flow structures.

8.2.1c. Discussion. From the results three main conclusions can be made; First, at the end of compression axial turbulence is amplified followed by an amplification of the other components. Second, the turbulent length scale is first
increasing and thereafter decreasing. Third, an increase in swirl number leads to increased turbulence dissipation.

Considering the first conclusion, the cause of high turbulence levels at the end of compression has been assumed to be converted from energy available in the tumble motion. This is likely to be true for engines with high tumble numbers such as those studied by Moreau et al. (2005), Fogleman et al. (2004) and Borée et al. (2002) among others. However, for the case of isotropic engine studied in the present work the increase in energy can only be originating from the piston. What has been noticed in the simulations is an increase in the vorticity perpendicular to the cylinder axis. This vorticity increase can be explained by combining the vortex stretching and vorticity dilatation interaction in the vorticity equation, Eqn. (5.2), which gives:

\[
\left( \vec{\omega} \cdot \vec{\nabla} \right) \vec{U} - \vec{\omega} \left( \vec{\nabla} \cdot \vec{U} \right) = \begin{pmatrix}
\omega_y \frac{\partial U}{\partial y} + \omega_z \frac{\partial U}{\partial z} - \omega_x \left( \frac{\partial V}{\partial y} + \frac{\partial W}{\partial z} \right) \\
\omega_x \frac{\partial V}{\partial x} + \omega_z \frac{\partial V}{\partial z} - \omega_y \left( \frac{\partial U}{\partial x} + \frac{\partial W}{\partial z} \right) \\
\omega_x \frac{\partial W}{\partial x} + \omega_y \frac{\partial W}{\partial y} - \omega_z \left( \frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} \right)
\end{pmatrix}
\]  

where the direct effects of compression are encircled. Hence, it can be seen that compression only increases vorticity perpendicular to the cylinder axis, which in turn increases the axial turbulent fluctuations. This amplification of vorticity is present even if no global elliptic instability can be found.

The second conclusion can likely by explained as; the increase in turbulent length scale noticed in the beginning of the compression in Fig. 8.14 is due to the dissipation of the smallest turbulent scales thus making the turbulent length scale statistically larger. When the turbulence levels start to increase,
8.2. RESULTS, EFFECTS OF COMPRESSION

![Diagram of turbulent fluctuations](image1)

**Figure 8.16.** Turbulent fluctuations, $(-- u'_{\vartheta} u'_{\vartheta})$, $(--- u'_{r} u'_{r})$, $(-- u'_{z} u'_{z})$. Rows correspond to swirl number 0 and 4, respectively (Söder et al. 2013a).

i.e. an increase in production, more kinetic energy is diffused to the smaller scales of the flow while the largest flow scales are constant in size.

The third conclusion is partly in agreement with literature of isotropic turbulence in swirling flows. Ibbetson & Tritton (1975) found that rotation in pipe flow increases turbulent decay due to wall interaction. It was also found that rotation increased the integral length scale parallel to the rotational axis. However, for rotational flow without wall interaction an increase in the rotation has been found to reduce dissipation, Bardina et al. (1985). Rotation was also found to redistribute energy in the wavenumber space leading to an increase in length scales in the direction of rotation. The reduction of dissipation presented in this thesis can be explained by wall interaction. However, the redistribution of energy in the wavenumber space could be an interesting topic for further research.
8. RESULTS AND DISCUSSION

8.2.2. Tilted rotating motion

First, the effect of tilt angle on flows, with the same total kinetic energy at BDC, undergoing compression are presented. Thereafter, the effect tilt has on mixing followed by the effects on the flow field at TDC are presented.

8.2.2a. Flow kinetic energy. With increasing tilt angle, Fig. 3.1, it can be seen that the peak kinetic energy becomes larger as well as appearing later during the compression stroke, Fig. 8.17. Generally, the total kinetic energy at TDC reduces with tilt angle. However, maximum total kinetic energy at TDC is found for low tilt angles. In Fig. 8.18 the kinetic energy of the swirl ($E_{\Omega,z}$) and tumble ($E_{\Omega,z}$) are depicted. It can, for the examined geometry (no piston bowl), be seen that the kinetic energy of the swirl is unaffected by compression. Tumble kinetic energy on the other hand is first amplified by the compression, but as the tumbling motion starts to break down around mid-stroke the energy is quickly reduced. This is as discussed in Sec. 3.2 known as tumble breakdown. For the examined geometry and rotational strength the kinetic energy of the tumble had completely broken down by the time of maximum dilatation (relative volume change). However, examining the turbulent kinetic energy for the $9^\circ$ tilt case we observed that the turbulence is increasing after this point, Fig. 8.19. This amplification is caused by vorticity-dilatation as discussed in Sec. 8.2.1c. Peak turbulence levels are found to increase with increasing tilt angle. However, at TDC it is found that highest turbulence level was obtained for the $61^\circ$ tilt case.

![Energy vs. CAD](image_url)

**Figure 8.17.** Evolution of total kinetic energy during compression. Vertical lines indicate, from left, mid-stroke, Cylinder height = radius, maximum dilatation and TDC, respectively (Söder et al. 2014).
8.2. RESULTS, EFFECTS OF COMPRESSION

**Figure 8.18.** Evolution of the two components of rotational kinetic energy during compression. Vertical lines indicate, from left, mid-stroke, Cylinder height = radius, maximum dilatation and TDC, respectively (Söder et al. 2014).

**Figure 8.19.** Evolution of turbulent kinetic energy during compression. Vertical lines indicate, from left, mid-stroke, Cylinder height = radius, maximum dilatation and TDC, respectively (Söder et al. 2014).
8. RESULTS AND DISCUSSION

8.2.2b. Mixing. If one tries to obtain a radially stratified EGR mixture it is important that the tilt angle is no greater than 9°, Fig. 8.20. As significant mixing was obtained for higher tilt angles even if the flow field is perfectly stratified at BDC.

8.2.2c. Flow at TDC. At TDC the flow was of solid body type for a radial position \( r \) greater than 60 % of the cylinder radius \( R \), Fig. 8.21. However, in the cylinder center a speed-up of the rotational motion for 9°, 19° and 34° tilt angles is observed. This deviation is significant even if caution regarding the smaller averaging area in the cylinder center is taken.
8.2. RESULTS, EFFECTS OF COMPRESSION

Figure 8.21. Distribution of angular velocity in the midplane at TDC, averaged at a specific radius. The very low value of $U_\omega$ (-28) is caused by the absence of mean angular velocity. Dashed lines indicate the mean angular velocity calculated using Eqn. (5.7) (Söder et al. 2014).

8.2.2d. Discussion. From the results of flows with different tilt angles the main conclusions and hypothesis that can be drawn are; First, a small tilt angle has a significant effect on angular velocity distribution and turbulence at TDC. Second, the peak turbulence is increasing with BDC tumble number. This is hypothesized to be caused by higher vortex stretching/tilting as compared to dilatation (which is constant). Third, a circumferentially homogeneous and radially stratified gas mixture is impossible for tilt angles of 19° or larger.
8. RESULTS AND DISCUSSION

8.2.3. Rotational strength (tumble number)

Note: In order to compare the different energy levels the results were scaled to have the same kinetic energy at BDC as the TN 1.23 case.

8.2.3a. Examining the relative importance of dilatation to tumble breakdown for different tumble numbers. By reducing the tumble number of the initial flow by half, the vortex stretching/tilting term in the vorticity equation, Eqn. (5.2), is reduced by a factor 4 while the vorticity-dilatation term is reduced by a factor of two. This can be used to study the relative importance of tumble breakdown (governed by stretching/tilting) to dilatation. It can be seen that with a reduction in tumble number the onset of tumble breakdown is delayed, see Fig. 8.22. This delay can be noticed in the relatively higher and later peak in tumble kinetic energy. Also, it can be observed that the tumble for all but the lowest tumble number case has broken down by the time of maximum dilatation. In Fig. 8.23 it can be seen that the scaled turbulence level is significantly increased with reduced tumble number. The peak turbulence level is found to be delayed with a reduction in tumble number, indicating that the relative importance of the dilatation is increased as compared to the tumble breakdown. The hypothesis presented in Sec. 8.2.2d that higher tilt increases the importance of vortex stretching/tilting appears to be correct.

Additionally, for tumble numbers between 0.6-1.23 the unscaled turbulence levels at TDC collapse, Fig. 8.24 while slightly higher and lower values are found for the high (1.85) and low (0.3) tumble cases, respectively. Showing that a drastic change in tumble number is necessary to influence the turbulence level at TDC. However, prior to TDC the effect of tumble on turbulence level is greater (for a flat piston geometry).
8.2. RESULTS, EFFECTS OF COMPRESSION

Figure 8.22. Evolution of the two scaled components of rotational kinetic energy during compression for pure tumbling motion of different strengths. Vertical lines indicate, from left, mid-stroke, Cylinder height = radius, maximum dilatation and TDC, respectively. The energy is scaled to correspond to the TN 1.23 case at BDC.

Figure 8.23. Evolution of scaled turbulent kinetic energy during compression. Vertical lines indicate, from left, mid-stroke, Cylinder height = radius, maximum dilatation and TDC, respectively. The energy is scaled to correspond to the TN 1.23 case at BDC.
Figure 8.24. Evolution of unscaled turbulent kinetic energy during compression. Vertical lines indicate, from left, mid-stroke, Cylinder height = radius, maximum dilatation and TDC, respectively.
8.2.4. Effect of the rotational strength of swirl and tumble

8.2.4a. The effect of swirl on turbulence for a tilted rotation. By comparing the results for a pure tumble case with the results for a tilted rotation with a similar tumble number, i.e. the setup used in Sec. 8.2.2 with Sec. 8.2.3, the effect swirl has on turbulence can be studied.

In Fig. 8.25 it can be seen that the tumble motion is amplified more for the cases without swirl as compared to the tilted case with the same tumble number. A similar trend can be seen for the turbulence level, Fig. 8.26. Therefore, swirl appears to have a dampening effect on turbulence even for a more complex flow field as compared to the flow field discussed in Sec. 8.2.1.

![Figure 8.25. Evolution of the rotational kinetic energy of the tumble motion during compression.](image1)

![Figure 8.26. Evolution of turbulent kinetic energy during compression.](image2)
8.2.4b. The effect of rotational strength and tilt angle on the flow. By plotting the peak turbulence level as a function of tumble number at BDC and tilt angle it can be seen that peak turbulence levels increase with BDC tumble number, Fig. 8.27a. The turbulence level at TDC does not exhibit a similar trend, instead the highest turbulence level at TDC is found for the 61° tilt case with intermediate BDC tumble, see Fig. 8.27b. In Fig. 8.27c the effect of BDC tumble number and tilt on the turbulence level 10 crank angles prior to TDC is shown. It is observed that the effect of tumble and tilt on turbulence level is significantly greater prior to TDC as compared to at TDC. This is caused by two effects; first, increasing tilt and/or tumble increases peak turbulence level; second, increasing tilt increases turbulent dissipation. It is hypothesized that the pure tumble case breaks down into smaller turbulent structures as compared to the tilted case, which in turn leads to higher dissipation.

During compression, the swirl number is found to be reduced between 4 and 6% due to viscous dissipation, Fig. 8.27d. If the tilt angle is increased the dissipation of swirl is increased. Similarly increasing the swirl number at BDC appears to increase the dissipation. This change (although small) is hypothesized to be caused by two effects. First, increasing the tilt increases turbulence which should lead to higher losses. Second, increasing the swirl leads to stronger secondary motions which has a dissipative effect on swirl.

Total kinetic energy at TDC is (not surprisingly) increasing with increasing total kinetic energy at BDC and, generally, is reduced with increased tilt, Fig. 8.27e. The reduction with tilt angle is caused by the higher peak turbulence level associated with higher tilt angle.

In order to keep the passive scalar $C1$ radially stratified ($UI<0.6$) it is important to keep the tilt angle low, Fig. 8.27f. Keeping the tilt angle low leads to lower turbulence levels which in turn leads to less mixing.
8.2. RESULTS, EFFECTS OF COMPRESSION

Figure 8.27. (a) Peak turbulence level. (b) TDC turbulence level. (c) Turbulence level 10 CAD angles prior to TDC. (d) Percentage loss of swirl BDC to TDC. (e) Total kinetic energy at TDC. (f) Uniformity at TDC. As a function of tilt angle and BDC swirl/tumble or kinetic energy. Interpolated surface from calculated cases (red dots).
8.3. Results, Piston motion

8.3.1. Overview

During the intake, there is first a rapid increase of tumble, while swirl is decreased, see Fig. 8.28. The reduction of swirl is not due to loss of angular momentum but rather due to the increase of in-cylinder mass. Around crank angle 30, swirl starts to increase rapidly while tumble reaches a plateau between crank angle 50 and 120. Tumble is thereafter reduced due to the rapidly increasing moment of inertia.

![Swirl and tumble during intake - exhaust.](image)

8.3.2. Error estimate of the Thien swirl number

By running a motored engine with fixed valve lift, it is possible to compare a Thien swirl number calculated from the swirl coefficient of a single valve lift in a steady swirl rig geometry to what can be expected in an engine. First, it can be observed that estimating mass flow using the piston motion is a relatively good approximation of the mass flow, Fig. 8.29a. However, the estimated momentum flow differs significantly between the cases, Fig. 8.29b. It can be observed that the momentum flow into the cylinder is much later as compared to the momentum flow estimated using the Thien methodology. Again, this is partly due to the time it takes to empty the volume above the ports, Fig. 8.9 (dashed purple line), and partly due to the effect of fluid acceleration/deceleration on the swirl coefficient as described in Sec. 8.1.1. Overall, this leads to an overestimation of the swirl number of more than 10% using the Thien methodology. A 10% increase of the swirl number can be expected to increase the NOx emissions with 4% (Thundil Karuppa Raj & Manimaran (2012)) and reduce
soot emissions between 5 % and 50 %, see Thundil Karuppa Raj & Manimaran (2012) and Bonatesta et al. (2007), respectively.

8.3.3. Improving the swirl Number estimate

By setting the swirl coefficient to zero during the time it takes for the volume above the valves to enter the cylinder, Fig. 8.9, a better agreement between the in-cylinder momentum (swirl number) and estimated swirl number, Fig. 8.30. This methodology is henceforth called displacement improved Thien swirl ($SN_{DIT}$) number and reduces the overestimation of swirl from 11 % to 7.5 %. During compression, the swirl number was found to reduce by 4-6 %, see Fig.8.27d. Similar reduction can be expected during intake, in Fig. 8.30 the reduction can be estimated to approximately 4 % (From 1.8 to 1.73). Therefore, it is suggested that calculated $SN_{DIT}$ is reduced by an additional 5 % before used as initial condition in sector model simulations.

8.3.4. CCV for a motored engine with fixed valves

For the swirl dominated engine studied it was found that the cycle-to-cycle variation (CCV) in tumble is relatively greater than the CCV in swirl, Fig. 8.31. The CCV in tumble leads to significant difference in peak turbulence between cycles, Fig. 8.32. The is due to the strong influence pf tumble on turbulence as described earlier, Sec. 8.2.4b.
8. RESULTS AND DISCUSSION

Figure 8.30. Cumulative swirl number, using the momentum flow for the piston case, steady case, using the swirl coefficient from the steady case with Thien methodology ($SN_T$), displacement improved Thien swirl number (designated $SN_{DT}$) and using the angular momentum within the cylinder for the piston case, Black solid, dotted, dash dotted, dashed and solid green line, respectively.

Figure 8.31. Swirl and tumble for the different cycles, solid and dashed line, respectively.
8.3.5. Effect of engine speed

Generally, engine speed is not considered to affect neither in-cylinder mass nor swirl/tumble. However, by changing the engine speed in the simulations from 1200 rpm to 1800 rpm, the in-cylinder mass increased, Fig. 8.33. The effect can be seen in the end of the intake stroke and is caused by the flow inertia. At the beginning of the intake stroke the cylinder volume is small leading to a small phase lag between pressure and mass flow, i.e. the air within the ports are accelerated irrespectively of the engine speed. However, at the end of the intake stroke the cylinder volume has increased significantly leading to an increased phase lag. This lag can be explained by the time it takes a pressure pulse traveling from the piston to the cylinder head, which is approximately 3 and 5 crank angles for the 1200 and 1800 rpm cases, respectively (assuming the speed of sound to be 340 m/s). In Fig. 8.34 the effect on swirl and tumble is shown. It is noted that tumble appears to be more sensitive to engine speed than swirl.
8. RESULTS AND DISCUSSION

Figure 8.33. The effect of engine speed on in-cylinder mass. 1200 and 1800 rpm solid black and dashed blue line, respectively.

Figure 8.34. The effect of engine speed (1200 and 1800 rpm solid and dashed line, respectively) on swirl and tumble, black and blue line, respectively.
CHAPTER 9

Summary and conclusions

9.1. Dynamic effects during intake

PIV measurements and LES computations were performed on a steady swirl test rig geometry. Three different valve lifts and driving pressures were used in the measurements. During the intake, the gas is given angular momentum when flowing through the intake ports. It is observed that higher valve lift increases in-cylinder swirl level and decreases turbulence. Fluctuations in swirl coefficient at low valve lifts are found to be greater compared to the averaged swirl coefficient. Therefore, swirl fluctuations may be of greater importance than the mean swirl coefficient for low valve lifts.

By varying the boundary conditions in the steady swirl test rig a phase lag between the mass flow at the outlet and the mass flow entering the cylinder leading to an overshoot. The phase lag was observed to be responsible for a swirl coefficient dependency on flow acceleration/deceleration. In addition, the volume above the valves was found to have a significant effect on early intake stroke angular momentum flow and thus on swirl number.

By moving the valves in a geometry otherwise similar to a steady swirl test rig it was possible to examine the quasi-steady assumption generally applied during engine design. It was noticed that the valve motion has a non-negligible effect, as higher mass and angular momentum flow were observed during valve closing as compared to during valve opening. These differences were attributed to hysteresis and pressure gradient created by the valve acceleration.

The results can be summarized as follows:

- Driving pressure has no effect on neither mean flow field nor turbulent quantities, for steady flow.
- At low valve lifts, the swirl coefficient measured in a steady swirl test rig setup is insufficient to characterize the flow.
- With increasing valve lifts the swirl coefficient increases and the fluctuations in the swirl coefficient decreases.
- Turbulence is found to be more anisotropic at low valve lifts.
- Flow acceleration reduces the swirl coefficient.
- Flow deceleration increases the swirl coefficient.
9. SUMMARY AND CONCLUSIONS

- The swirl coefficient was observed to be higher during valve closing as compared to valve opening.
- Mass flow is higher during valve closing as compared to the same lift during valve opening. This difference is likely caused by flow inertia, which leads to a phase lag between pressure and velocity.
- During early intake, no angular momentum enters the cylinder until the volume above the valves has been emptied.

9.2. Effects of compression

Large eddy simulations were performed on a simplified engine geometry with different initial conditions. The initialized flow fields were compressed and the effects were studied. Compression was found to increase vorticity perpendicular to the cylinder axis through vorticity-dilatation. This in turn increases turbulent fluctuations in the cylinder axis.

Swirl was found to decrease turbulent fluctuations for both a pure swirl with initially isotropic turbulence as well as for cases with a tilted rotation. From a literature study it is believed that the decay of turbulence is due to wall interaction. However, swirl is also known to redistribute kinetic energy in the wavenumber space and this may play a role for swirling flow under compression.

- Swirl reduces in-cylinder turbulence during compression.
- A tilted rotational motion at BDC leads to a large deviation from solid-body rotation at TDC.
- Compression increases axial turbulent fluctuations.
- Increased tilt leads to higher and earlier peak turbulence level.
- Peak turbulence level was found to increase with the tumble number at BDC.
- TDC turbulence level was observed to be relatively insensitive to tilt or tumble number. However, ten crank angles prior to TDC the difference is significant.
- Increased turbulence increases mixing, i.e. higher tilt or tumble number leads to better in-cylinder mixing.
- Highest kinetic energy at TDC was found for low but non-zero tilt angles.
- During compression the transverse length scale of tangential velocity fluctuations is found to increase during early compression and decreasing at the end of compression.
- In order to maintain a radially stratified gas mixture during compression it is important to keep the tilt angle small. With increasing BDC swirl number a lower tilt angle is needed to keep the uniformity index constant. Additionally, coupled with the results presented by Miles (2009) (described in Sec. 3.3) the BDC swirl number has to be high enough to...
keep the squish flow attached to the piston wall if an EGR flow at the high temperature areas are sought.

9.3. Piston motion

Large eddy simulations were performed for a motored heavy-duty diesel engine without piston bowl and with fixed valves; the valves were closed instantaneously at BDC. The used geometry made it possible to validate the proposed methodology for estimating turbulence. Also, it was possible to give an error estimate for the Thien swirl number and improvements are proposed. Additionally, the effect of engine speed on in-cylinder mass and angular momentum was studied.

- Larger (relative) fluctuations was observed for tumble as compared to swirl.
- The methodology of calculating turbulence was found to be in good agreement with turbulence calculated using a phase-average.
- By setting the swirl coefficient to zero until the "undirected port volume" is emptied, a more accurate swirl number can be obtained.
- To further increase the agreement between swirl number calculated from flow of angular momentum and swirl number obtained at BDC, the angular momentum entering the cylinder was reduced by 5%. This reduction was chosen as it was the approximate loss of angular momentum during compression.
- A moderate (increasing) effect of engine speed on in-cylinder mass and tumble were observed.

9.4. Suggested implementation of the results during engine design

The aim of the work presented in this thesis was to enable an increase the fuel efficiency of heavy-duty diesel engines by increasing the understanding of the physical processes during intake and compression. In the previous chapters the conclusions drawn from the simulations were presented. In this chapter, it is presented how these conclusions can be used to improve engine efficiency. As discussed in Sec. 3.5.4, engine optimization is commonly performed using sector models with a RANS turbulence model. Two of main assumptions behind the forementioned methodology are that the swirl number used as initial condition is the swirl found in a real engine and that the rotation is a pure solid body rotation.

9.5. Correct swirl number

In the presented work several problems regarding how the swirl number is calculated have been observed. An improved method of calculating the swirl number, which will improve the agreement between calculated swirl number
and the swirl experienced inside the engine has been proposed and should be used. This method should be further improved to account for the residual gas, compressible effects (engine speed), flow acceleration/deceleration and valve motion. At low valve lifts, it was found that the fluctuation in swirl was significant, meaning that the swirl angular momentum created at low valve lifts can vary significantly between cycles. Therefore, it is recommended that a sensitivity analysis is preformed to observe possible effects during engine design.

9.6. Effect of non-zero tilt

A small tilt angle was found to increase the angular velocity in the center of the cylinder. Therefore, the effect of this speed-up should be investigated and studied separately to the speed-up caused by the reduction of moment of inertia if the piston has a bowl. A small tilt was also found to increase turbulence and mixing significantly. Therefore, keeping the tilt low is extremely important if radially stratified EGR is requested.

9.7. Implications for SI engines

It was found that highest turbulence level at TDC was obtained for a slightly tilted rotational motion. It was also hypothesized that this was caused by a larger turbulent length scale for a slightly tilted rotational motion. Therefore, it is suggested that different tilt angles are investigated during engine design.
CHAPTER 10

Self criticism and model uncertainties

*Anyone who has never made a mistake has never tried anything new.*

Albert Einstein

In order to obtain relevant results for complex systems, simplifications and approximations have to be made. Generally, these are known by the author but not always properly discussed. It may be in the explicit/implicit assumptions made regarding methodology and/or flow conditions. Criticism given by reviewers during the review process may also be of interest to the reader. This section is dedicated to share critique known by the author and of possible value to the reader. This section is written as questions answered by the author.

10.1. General comments

- SGS viscosity was modeled implicitly, i.e. the implicit LES methodology was used. How can this be justified?
  - First, a general requirement for a good LES simulation is that the effect of SGS viscosity is small, the effect of excluding the SGS viscosity should therefore by definition be small. Second, according to Celik *et al.* (2009) the size of numerical viscosity is of the same order of magnitude as the SGS viscosity. Therefore, adding SGS viscosity is likely to overestimate the total viscosity. Third, as discussed in Sec. 4.3.2c and the fact that the in-cylinder SGS turbulence are unlikely to be fully developed, the assumptions behind SGS models are not fulfilled leading to unknown errors.

- In all simulations, a first order temporal scheme has been used. How can this be motivated?
  - As discussed in Sec. 4.3.4c the PISO algorithm increases the temporal accuracy by one order, Issa (1986). Therefore, the Courant numbers used should be sufficient for the diffusion due to spatial discretization to be dominating. The Courant number was 0.2 for the compression cases while 0.7 during the intake. During intake,
the Courant number differs significantly within the domain, why the Courant number was well below 0.01 for most of the domain. This is important as a TVD scheme for the convective fluxes goes to a pure upwind scheme as the Courant number goes to 1, see Jasak (1996). Therefore, the low Courant numbers is more important to keep the accuracy of convective terms as compared to the temporal terms.

10.2. Dynamic effects during intake

- In the PIV experimental setup the swirling motion will enter the stagnation box 0.7 diameters below the measurement plane. How can this be expected to influence the results?
  - It is likely that precessing vortex core (PVC) will form. A PVC will lead to fluctuations in the swirl center. Therefore, a steeper velocity gradient can be expected to form within a real engine as compare to what can be seen in Fig. 8.3. In addition, the turbulent fluctuations are likely to be overestimated in the setup as compared to a real engine geometry. However, it is unlikely to change any of the conclusions drawn from the experiments.
- For the LES simulations, the boundary layer was neither resolved nor modeled using a wall model. How would this influence the results and conclusions?
  - In order to resolve the boundary layer the first cell center should be around $y^+ = 1$. This would ($U_\infty = 40$ m/s, $\rho = 1.2$ kg/m$^3$, $\mu=1.8 \cdot 10^{-5}$ kg/ms and $L=0.1$ m) require a cell height in the order of $6 \cdot 10^{-6}$ m. Not only practically impossible but also significantly smaller than the surface roughness, which (for cast iron) is in the order of $250 \cdot 10^{-6}$ m.
  - Wall models are designed for a fully turbulent boundary layer over a flat plate and the farther one deviates from this the greater error can be expected. In the studied geometry the port length is too short for a fully developed boundary layer to develop, large adverse pressure gradients and separations are also present further deviating from the theory behind wall models.
  - To sum up, resolving the boundary layer is not practically possible and a wall model is likely to overestimate the boundary layer thickness. However, none of the conclusions are likely to be affected if a wall model was used.
- When the valve motion was simulated coarser grids were used as compared to the other simulations. Why was this and what are the possible effects?
10.3. EFFECT OF COMPRESSION

The coarser grids were chosen entirely due to practical reasons (simulation time). The coarser grids could give quantitatively incorrect values, however, the difference between valve opening and closing is significant and is likely not to be caused by the grid size. A RANS simulation would likely lead to the same conclusions and should in retrospect been preferred. Additionally, as discussed in literature interesting features can be observed using coarse grid, Celik et al. (2005).

For the simulations regarding valve motion, both volume flow and swirl coefficient was greater during valve closing as compared to during valve opening. Could the difference in swirl coefficient be caused by the difference in volume flow?

This is one major drawback of the simulations, as one has to choose between constant pressure drop and constant volume flow. Constant pressure drop was considered more relevant why it was chosen. From the simulations, it is clear that a difference in volume flow between valve opening and closing exist. However, it is not possible to be certain how large the difference is and how a piston would affect the flow.

In a real engine the cylinder volume is smaller, and changing, compared to the volume used in the simulations. A smaller volume is likely to affect the in-cylinder pressure. Therefore, the presented simulations should be taken as a first indication that the valve motion has a significant effect, but more simulations are necessary.

10.3. Effect of compression

From Fig. 6.5 it is clear that the grid is finer in the center of the cylinder. How can this affect the results?

Generally, a finer grid leads to lower numerical diffusion, the numerical diffusion of turbulence in the outer part of the cylinder is thus likely to be higher as compared to the diffusion in the center of the cylinder. However, from Fig. 7.4 the diffusion appears to be low enough for this to have a secondary effect. Nevertheless, it should be considered, especially for Fig. 7 in Paper 5.

It was found that an increase in swirl lead to an increase in turbulent dissipation and according to Ibbetson & Tritton (1975) this might be caused by wall interaction, yet the boundary layer was neither resolved nor modeled. What are the possible effects of this?

The fact turbulent dissipation is noted indicates that the resolution is adequate. However, it would be interesting to compare the simulation with simulation where the boundary layer is resolved and with simulation with slip-wall boundary condition. A change
in wall resolution and boundary condition will not affect any of the conclusions drawn but rather increase the understanding of the mechanism behind the reduction in turbulence level.

- For the passive scalars a constant Schmidt number was used, Sec. 4.1.1. Yet the Schmidt number is dependent on viscosity, molecular diffusivity and density, all of which are changing during compression, see Eqn. 4.9. What implications do this have on the results?
  - Assuming that the BDC Schmidt number is 0.9, the molecular diffusivity follows the Chapman-Enskog Theory, constant collision integral, adiabatic walls and compression, the TDC Schmidt number will become 1.1. A simulation with an infinite Schmidt number was performed and compared to the original, no effect was observed. Therefore, the convective term is dominating the advection-diffusion equation and the approximation will not affect the results.

- How representative is one time step for a non-linear process as the one shown in Fig. 8.21.
  - The flow in an engine is highly non-linear and looking at a single time step for a single case is unlikely to be representative. However, as all cases 9°-61° (the 90° case is excluded due to division by zero) show the same trend, the results are likely to be representative. If one of the cases had shown a slowdown in the cylinder center, it would have been impossible to draw any conclusions.

10.4. Piston motion

- According to literature, 25 cycles is necessary for a converged mean velocity, see Sec.3.6, yet only 17 cycles were calculated at 1200 rpm and 5 cycles were calculated at 1800 rpm.
  - Indeed, if well-converged mean values are sought more cycles should be simulated. However, for the conclusions drawn the need of converged values is of lesser importance and when comparing 1200 and 1800 rpm the same trend (higher mass and tumble at 1800 rpm) can be seen for all individual cycles. Note that no conclusions of the type "an increase in engine speed from 1200 to 1800 rpm, leads to an increase in in-cylinder mass of XX %" were made.

- Very coarse grids were used during expansion and exhaust, how can this be expected to influence the results?
  - Coarser grids leads to more numerical dissipation, we can thus expected that the turbulence level in the cylinder at exhaust valve closing is lower then if finer grids had been used. A lower turbulence level may lead to a small reduction in cycle-to-cycle variations. However, the simulated engine was motored and it is
likely that introducing combustion would lead to a greater change in cycle-to-cycle variations. Again, no claims of exact values in cycle-to-cycle variations are made.
CHAPTER 11

Proposed future work

*Prediction is very difficult, especially about the future.*

Niels Bohr

The aim of all work presented in this thesis was to increase the knowledge of in-cylinder flows which in turn has the potential to increase engine efficiency. However, several steps need to be studied further to properly understand the flow structures created during intake and the effect of compression. It was wisely stated ages ago that:

*The more you know, the more you know you don’t know.*

Aristotle

During the course of the project, new known unknowns have risen. Regarding the intake flow, the exact cause of the difference in swirl coefficient between valve opening and valve closing needs more work to be properly understood. During compression, it was found that swirl reduces turbulence, it was hypothesized that this reduction was caused by wall effects. However, this remains to be proven and simulations with slip-wall boundary conditions are proposed to obtain further insight. A small tilt angle was found to increase the angular velocity in the cylinder center, no clear motivation why this was the case could be presented. The turbulent diffusion was observed to be significantly higher for a pure tumble case as compared to a case with 61° tilt angle. It is possible that a pure tumble breaks down into smaller turbulent structures as compared to the tilted cases, which would explain the higher dissipation. A POD analysis of the tumble breakdown for the different cases may provide additional insight into the breakdown process. Turbulence level was noted to have a maximum for a tilt angle of 61° and a BDC tumble number of 0.89, is this a generally the case or does it dependent on compression ratio, engine size and turbulence level at BDC?

The proposed future work can be summarized as follows:

- In the thesis, several hypotheses are presented, these needs to be validated/rejected. The presented hypotheses are:
11. PROPOSED FUTURE WORK

- With increasing tilt, the relative importance of vortex stretching/tilting (tumble breakdown) compared to vorticity-dilatation is increased.
- A tilted rotational motion breaks down into large scale turbulent structures as compared to a pure tumble which breaks down into small scale turbulence. Investigate how the turbulent length scales are affected by compression and tilt angle.
- Swirl reduces turbulence due to wall effects. Investigate the wall effects on dissipation of turbulence for different swirling flows. Preferably numerically where slip boundary conditions are possible coupled with simulations with a resolved boundary layer.

- Look into the effect of changed compression ratio, tilt and rotational strength on turbulence and mixing.
- Calculate/measure the slope of the turbulent spectra during tumble breakdown.
- Continue improving the methodology used to calculate the swirl number.
- Investigate why a higher swirl coefficient is obtained during valve closing as compared to during valve opening.
- Investigate how an off-centered rotational motion can be centered during compression.
- Bensler et al. (2002) hypothesized that minimizing swirl fluctuations in CI engines measured in the swirl test rig would reduce cycle-to-cycle variations. However, this hypothesis is yet to be proven.
- Study the relative effect of squish as compared to tilt angle on mixing and turbulence.
CHAPTER 12

Papers and authors contributions

**Paper 1**
*A Coupled PIV-LES Approach to Understand Port Generated Structures*
Martin Söder, Julie Vernet, Björn Lindgren & Laszlo Fuchs, 2010, in proceedings *LES4ICE*

In this paper, the flow in a steady swirl test rig was studied using PIV measurements and LES simulations. LES simulations were performed by Söder under supervision by Fuchs. PIV measurements were performed by Vernet under supervision by Söder and Lindgren. Steady swirl rig torque measurements were performed by Lindgren. The results were presented at International Conference on LES for Internal Combustion Engine Flows (LES4ICE), Paris France 2012 by Söder.

**Paper 2**
*Compression of a swirling and tumbling flow*
Martin Söder, Lisa Prahl Wittberg & Laszlo Fuchs, 2013, in proceedings *ASME ICEF2013*

In this paper a the effect of compression on a swirling and tumbling flow is studied in order to improve the understanding of turbulence production by compression. The work was performed by Söder under supervision by Prahl Wittberg and Fuchs. In proceedings of ASME Internal Combustion Engine Division Fall Technical Conference (ICEF) 2013.

**Paper 3**
*Effects of compression on coherent structures in an enclosure*
Martin Söder, Lisa Prahl Wittberg & Laszlo Fuchs, 2013, in proceedings *ICJWSF 2013*

98
The effect of swirl ratio and compression on isotropic turbulence is studied. The work was performed by Söder under supervision by Prahl Wittberg and Fuchs. The results were presented at 4th International Conference on Jets, Wakes and Separated Flows (ICJWSF), Nagoya Japan 2013 by Söder.

**Paper 4**

*Study of flow generated by the port in a heavy-duty diesel engine at different valve lifts using PIV*

Martin Söder, Julie Vernet, Björn Lindgren, Lisa Prahl Wittberg & Laszlo Fuchs,

This paper aim at increasing the understanding of the flow structures generated during intake using PIV measurements in a steady swirl test rig. PIV measurements were performed by Vernet under supervision by Söder and Lindgren. Post processing and analysis was performed by Söder and Vernet. Submitted to Experiments in Fluids.

**Paper 5**

*Effect of swirl/tumble (Tilt) angle on flow homogeneity, turbulence and mixing properties*


The effect of compression on tiltes rotational motions was studied. The work was performed by Söder under supervision by Prahl Wittberg and Fuchs. Lindgren contributed with comments regarding the industrial implications and proofreading. The results were presented at SAE 2014 International Powertrain, Fuels & Lubricants Meeting, Birmingham UK 2014 by Söder.

**Paper 6**

*Towards understanding the effect of compression on swirl and tumble in the context of turbulence and mixing*

Martin Söder, Lisa Prahl Wittberg & Laszlo Fuchs,

An extension of Paper 5. In this paper the effect of rotational strength was investigated. The work was performed by Söder under supervision by Prahl Wittberg and Fuchs. Submitted to Flow, Turbulence and Combustion

**Paper 7**

*Investigating the dynamic effects on flow structures generated during the intake*
stroke in heavy-duty diesel engines using Large Eddy Simulations
Martin Söder, Lisa Prahl Wittberg & Laszlo Fuchs,

By adding time varying boundary conditions, the dynamic effects during intake were studied. The work was performed by Söder under supervision by Prahl Wittberg and Fuchs.
Acknowledgements

In my walks, every man I meet is in some way my superior;
and in that I can learn of him
Ralph Waldo Emerson

First of all I would like to thank my supervisor Professor Laszlo Fuchs for all encouragement, tough questions and willingness to share his deep knowledge of fluid dynamics. My second supervisor Lisa Prahl Wittberg is acknowledged for her many comments on my articles. Björn Lindgren is acknowledged for proofreading an early version of this thesis and giving valuable discussion. I would also like to thank the other members of the project steering committee Niklas Nordin, Dennis Konstanzer, Raymond Reinmann, Stephen Conway, Jonas Holmborn and Per Lange for their support and input.

Furthermore, the financial support from Scania CV AB and the Swedish Energy Agency is greatly acknowledged. Special thanks to the CCGEx competence center for input and comments.

I would thank all my colleagues at Scania for a good and friendly environment. A special thanks to my former and present colleagues at NMTD, Raymond, Stephen, Niklas, Eric², Hannan, Kim, Micael, Zemichael, Magnus, Dennis and Thomas.

Thank you Johan Fjällman for a great environment in our office. A big thanks to my newer office mates Elias, Bertrand and Shyang, it was fun getting to know you! I would like to thank Olle Bodin, Emma Alenius, Alexander Sakowitz and Niklas Winkler for very rewarding discussions regarding methodology and of course for all the great times. Everyone of the Laszlo group is greatly thank for keeping the Monday meetings hard but forgiving.

All former and present graduate student of the lab (Mathias, Karl, Shintaro, Julie, Renzo, Marco, Tomas, Andreas, Alexandra, Ellinor, Marcus W,
Marcus P, Fredrik, Ramin, Sembian, Sissy, Ylva, Shahab, Malte and Outi) as well as all the seniors (Henrik, Nils, Jens, Bengt, Fredrik, Antonio) are acknowledged for their contribution to a great working environment. At Osquars Backe I would like to thank Stevin, Feng, Yue and Mireia for the time we shared corridor and office but also Bernhard and Alexander for great coffee and discussions when I have visited. Henrik Alfredsson is greatly acknowledged for the thorough reading of the thesis and providing good comments.

Petter Tornehed is gratefully thanked for his comments on my Licentiate thesis, moral support and nice lunches.

During the project I had the great opportunity to visit the energy department at Politecnico di Milano and I would like to thank everyone at the department for their hospitality and discussions. Thank you I learned immensely.

I would like to thank my parents, Ove and Åsa for their never-ending support. Almost last, but definitely not least, I would like to thank my wife Annika for love, support and proofreading my abstract.

At last, a huge thanks to the smallest, thank you Adrian for always greeting me with a smile!
References


REFERENCES


REFERENCES


IPCC 2007 *Climate change 2007: The physical science basis*. Cambridge University Press.


REFERENCES


PIERREHUMBERT, R. T. 2014 Climate Science Is Settled Enough. URL http://
REFERENCES


TOLEDO, M. S., LE PENVEN, L., BUFFAT, M., CADIOU, A. & PADILLA, J. 2007


Vernet, J. 2012 Detailed study of steady in-cylinder flow and turbulence using stereo-PIV. Master thesis, KTH.


Part II
Papers