Numerical Investigation of Internal Combustion Engine Related Flows

by

Martin Söder

June 2013
Technical Reports from
Royal Institute of Technology
KTH Mechanics
SE-100 44 Stockholm, Sweden
Akademisk avhandling som med tillstånd av Kungliga Tekniska Högskolan i Stockholm framläggdes till offentlig granskning för avläggande av licentiatexamen tisdagen den 20 augusti 2013 kl 10.15 i E2, Lindstedsvägen 3, Kungliga Tekniska Högskolan, Stockholm.

©Martin Söder
Universitetsservice US-AB, Stockholm 2013
Internal combustion engines has been used for more than 100 years. The use of the abundant energy supply stored as hydrocarbon fueled unprecedented economic growth. The use of hydrocarbons increased the work output of human labor significantly, thus increasing the economy and prosperity. However, during the latter part of the twentieth century negative consequences of the internal combustion engine has been noticed. Initially the being emissions of local pollutants such as carbon monoxide, nitrogen oxides and unburnt hydrocarbons. These pollutants have to this day in the western world been reduced significantly and further reductions are under way. Thereafter, has the focus been shifted somewhat to global emissions such as carbon dioxide due to the effect on the climate. However, as the most accessible oil resources have been exhausted the price of oil has five folded since the turn of the century, straining the exponential economic growth enjoyed for two centuries.

Heavy duty diesel engine efficiency is still below 50%, there is thus a need and a possibility to further increase engine efficiency. In this thesis, work has been done to increase the understanding of the flow prior to combustion. A better knowledge of pre-combustion in-cylinder flow would increase the possibility to reduce engine emissions and fuel consumption, through better mixing and lower heat transfer.

The work presented is ordered in such a way that the flow structures created during the intake is presented first. Thereafter, the effect of compression is investigated. Intake flow structures are studied using Large-Eddy Simulations (LES) and experiments on a steady swirl test rig. The effects of compression are studied using simulations of predefined flow structures undergoing compression.

It is found that the flow structures created during intake is qualitatively different depending of intake valve lift. And that a single Swirl Number (SN) is an insufficient quantity to characterize the flow created at low valve lifts, due to high fluctuations. During compression it is found that a high swirl number suppress small scale turbulence while the compression has an increasing effect of axial fluctuations due to vorticity-dilation interaction. Additionally, it is shown that turbulent kinetic energy is introduced in the flow field by the piston in the absence of tumble breakdown.

Descriptors: In-cylinder flow structures, Engine turbulence, Engine simulations, Large Eddy Simulation.
Preface

This Licentiate thesis consists of two parts. The first part gives an introduction to engine emission formation, in-cylinder flow and computational aspects. The second part consists of four papers. The work presented in this thesis has been performed by the author under supervision of Dr. Lisa Prahl Wittberg and Professor Laszlo Fuchs.

June 2013, Stockholm

Martin Söder
Ignorance more frequently begets confidence than does knowledge.

Charles Darwin
## Contents

Abstract iii

Preface iv

Chapter 1. Introduction 1

Chapter 2. Internal Combustion Engines 5
  2.1. IC engine cycle 5
  2.2. Emissions from Diesel Combustion 6
  2.3. Emission Legislation 8
  2.4. The Combustion Engine; History, Energy and the economy 9

Chapter 3. Flow of Internal Combustion Engines 12
  3.1. Swirl and Tumble 12
  3.2. Squish flow 14
  3.3. In-cylinder Turbulence 15

Chapter 4. Models and Methods 17
  4.1. Governing equations 17
  4.2. Vorticity 18
  4.3. Kinetic energy 19
  4.4. Turbulent anisotropy 20
  4.5. Proper Orthogonal Decomposition 21
  4.6. Particle Image Velocimetry 23
  4.7. Computational Fluid Dynamics 24

Chapter 5. Flow under consideration 29
  5.1. Steady flow rig 29
  5.2. Effects of compression 32

Chapter 6. Results and discussion 35
| Chapter 6. | Results, steady flow rig | 35 |
| Chapter 6. | Results, Effects of compression | 47 |
| Chapter 7. | Summary and conclusions | 57 |
| Chapter 7. | Conclusions, steady flow rig | 57 |
| Chapter 7. | Conclusions, Effects of compression | 57 |
| Chapter 8. | Future work | 59 |
| Chapter 9. | Papers and authors contributions | 61 |
| Acknowledgements | 64 |
| References | 65 |
| Paper 1. | A Coupled PIV-LES Approach to Understand Port Generated Structures | 72 |
| Paper 2. | Study of in-cylinder flow structures using PIV and LES | 92 |
| Paper 3. | Compression of a swirling and tumbling flow | 130 |
| Paper 4. | Effects of compression on coherent structures in an enclosure | 152 |
Part I
Overview and summary
CHAPTER 1

Introduction

Emissions from vehicles have been a great health problem in many cities, among others Los Angeles that due to its location is especially smog prone. Smog is a serious health hazard which has led to an increase in emission legislation, EPA (1998). In Europe the current Euro legislation started in 1992 with the Euro I legislation regulating nitrogen oxides and particle mass per kilowatt-hour of work. The regulation has since become stricter and in January 2014 all new trucks have to conform to the Euro VI legislation, dictating not only nitrogen oxides and particle mass but also number of particles.

Different strategies have been used to meet these emission targets such as injection timing, Selective Catalytic Reduction (SCR) and Exhaust Gas Recirculation (EGR) to reduce nitrogen oxide emissions and filters and increased injection pressure to reduce particle mass. Generally, these strategies lead to increased fuel consumption and/or cost. Lower fuel consumption is for the transporting sector equivalent to lower cost. Therefore, lower fuel consumption is sought by all heavy duty engine manufactures due to the fierce competition. Fuel consumption is also 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), and further increases in oil production is relying on more expensive unconventional sources, high oil prices are likely to stay. The effect of high oil prices and an increasing concern of climate change are likely to increase the effort of reducing fuel consumption.

In order to reduce fuel consumption without increasing emissions, better knowledge of all physical processes are necessary. This work is focused on fluid motion in the cylinder of heavy duty Diesel engines prior to injection. After treatment systems such as SCR and filters will thus not be examined in this project.

Increased understanding in the pre-combustion mixing process will enable engine manufactures to reduce the amount of exhaust gases used and increase oxidation of soot. In order to optimize the use of EGR, the exhaust gases should either be perfectly mixed or stratified in such a manner that the highest concentrations are located at the position of peak combustion temperature.
Generally, post combustion some unburnt fuel residues will form soot. In order to minimize soot emissions it is desirable to mix fuel residues with oxygen rich parts of the post combustion mixture. Therefore, knowledge of pre-combustion flow structures and coupling with post-combustion flow field is interesting in reducing emissions.

During intake stroke the large-scale motions in the engine cylinder are created. The largest structures are called swirl and tumble, where swirl is the flow angular velocity around the cylinder axis, often divided by the angular velocity of the crank shaft, $\omega_E$, i.e. the swirl number. Tumble is flow angular velocity around an axis perpendicular to the cylinder axis.

Formation of structures created during intake has historically been studied using steady swirl test rigs. These rigs are used to measure the flux of angular momentum around the cylinder axis created at different valve lifts. A swirl number is then calculated from the measured angular momentum flux and used to characterize the engine. 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 of 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. It was found that for spark ignition engines the best flow characteristics measurable using PIV in a steady flow rig were tumble and overall flow kinetic energy. Historically, steady swirl test rig measurements have worked relatively well to characterize the engine but with the increasing demand deeper knowledge of the created structures and the effect of compression is necessary.

During compression the angular momentum of the swirl is preserved. However, the angular momentum of the tumble motion breaks down during compression, Lumley (1999). How these large-scale structures interact in creating the flow field at Top Dead Center (TDC) is not completely understood. Especially the cause of the tumble breakdown has been debated. In Lumley (2001) work Obukhov, among others, is discussed. Obukhov showed that rotation around the middle axis of inertia is unstable. Additionally, Gledzer & Ponomarev (1992) report instabilities around the major axis at certain axis ratios. Lundgren & Mansour (1995) showed that elliptic instabilities occur for vortices with aspect ratio of two. Yet no conclusive evidence has been shown for the flow within an engine.

In order to study the effect of compression on tumble, several numerical and experimental studies has been done on compressing a well-defined tumbling flow in a rectangular geometry to a compression ratio of four at 206 rpm. Boree et al. (2002) noticed that at a volumetric ratio of two, mean kinetic energy was transferred from the tumble motion to turbulence at a timescale in the order of vortex turnover time. It was also noticed that small deviations
1. INTRODUCTION

in the large scale motion (tumble) was amplified during compression. Moreau et al. (2004) injected fluid into the tumble motion and studied the flow using PIV measurements. They noticed that the injected air destabilized the vortex independently if the air increased or decreased the angular momentum of the flow, hypothesizing that turbulence expedite the breakdown of turbulence. Toledo et al. (2007) performed LES simulations and noticed that wall interaction during compression is important as instabilities were transported into the rotating flow. It was also observed that in the late part of compression the tumble trajectory becomes erratic and very dependent on initial condition.

Several studies have studied the flow in more realistic engine designs. Howarth & Jansen (2000) was the first to conduct Large-eddy simulation (LES) on engine geometry. Fogleman et al. (2004) saw an increase in turbulence around 30 to 60 crank angles before top dead center and assumed that this was caused by tumble breakdown. It was also noticed that a defined flow, i.e. clear tumble, reduced cycle to cycle variations. Studies of the effect of off-centered swirl on combustion include Dembinski & Ångström (2012), who noticed using PIV experiments, that off-centered swirl survives compression and has an effect on combustion. Similar observations have been done by Adomeit et al. (2006) who observed using RANS simulations and experiments that soot oxidation is strongly influenced (increased) by inhomogeneities of the swirl motion. According to Ge et al. (2008) off-centered swirl increases turbulence intensity which reduces soot. Contrary to reported by other authors (e.g. Adomeit et al. (2006)), it can however indicate that for the examined engine maximizing soot oxidation is more important than minimizing soot production. Rezaei et al. (2010) noticed using LES that eccentricity in the in cylinder swirl pattern affects soot production and oxidation, it was also noticed that different valve lifts strategies giving the same measured swirl number will impact combustion differently.

Yamato et al. (2001) found that fuel stratification is possible in a gasoline engine, using RANS simulations and PIV measurements. Buschbeck et al. (2012) noted, using PIV measurements on a Spark Ignition (SI) engine, that for stoichiometric condition the total flow kinetic energy is very important to reduce cycle to cycle variation. For lean operating conditions it is observed that the large scale motions close to the spark plug is more important than total kinetic energy. Squish has an important effect on the fluid motion in the end of compression. Miles (2009) showed that dependent on swirl number the squish flow either follows the cylinder head or is forced along the piston bowl. Although very important in the late part of compression it was noted by Payri et al. (2004), using RANS and LDV, that the piston bowl has very little effect on the creation of structures during intake. 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. According to Vermorel et al. (2009) low tumble
numbers increases cycle to cycle variations for SI engines, which is consistent to what was reported by Fogleman et al. (2004).

In order to perform LES simulations on in-cylinder flow extensive work has been done on necessary mesh and numerical methods. Where Moureau et al. (2004) noticed that first order mapping between different meshes reduces the accuracy of the simulations. Enaux et al. (2011b) notice that at least 25 cycles were necessary to get accurate mean values close to TDC, to obtain converged RMS values (less than 10%) at least 50 cycles is needed. It was also noticed that traditional Smagorinsky with low order (2nd) schemes are to dissipative. Celik et al. (2001) reviewed known 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. It was also reported that numerical grids with large aspect ratio (i.e. the ratio between the cell widths in different directions) will reduce the accuracy of the simulation significantly.

In this work the goal has been to first study the structures created during the intake using simulations on a steady swirl test rig. This was then compared to PIV and swirl test rig torque measurements. Furthermore, the effects of compression were studied on two kinds of characteristic flows using LES.

The main findings in this work is that turbulence levels found at TDC for swirling/tumbling flows is more dependent on kinetic energy introduced by the piston than on initial kinetic energy present in the swirling and tumbling motion. Moreover, the mean swirl number is shown not to be a relevant description of the engine flow structures at low valve lifts. At low lifts the anisotropy of the flow was also found to be much greater as compared to high valve lifts.
CHAPTER 2

Internal Combustion Engines

This chapter covers the basics regarding internal combustion engines and their effect on the environment. First, the internal combustion cycle is described and thereafter follows a description of the emissions and legislation thereof. Finally the energy and economic perspective is briefly discussed.

2.1. IC engine cycle

The main principle of the internal combustion engine is to convert chemical energy into mechanical energy. There are several ways in which this can be obtained, but the most common is the four-stroke engine.

2.1.1. Four stroke cycle

The four-stroke engine works in four distinct, although sometimes overlapping, phases:

1. Intake stroke (or induction stroke) The piston starts at TDC, moving downward while 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. At the end of the intake phase, close to bottom dead center (BDC), the valves close.

2. 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 is converted to potential energy in the form of pressure.

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

4. 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.

2.1.2. Combustion processes

For internal combustion engines, two different types of combustion processes are traditionally used; spark ignited combustion and compression ignition. 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 primarily uses Diesel fuel and is often called Diesel engines. The combustion processes in the two engine types are different. In the direct injected Diesel engine, only the fuel around the fuel spray has enough oxygen to burn (diffusion flame) whereas for the spark-ignited engine, a premixed combustion occurs. This work is focused on heavy duty Diesel engines, i.e. commercial trucks, although most results are probably applicable to other engine types as well.

2.2. Emissions from Diesel Combustion

Engine emissions can be divided into two subcategories; local and global. Local emissions, NO\textsubscript{x}, HC, SO\textsubscript{2}, CO and particles, have negative impact on the close surrounding, leading to health issues. Global emissions 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\textsubscript{2}, is tightly linked with fuel efficiency and is thus minimized for economic purposes. However, there is a trade-off between NO\textsubscript{x} emissions and combustion efficiency. Therefore, the introduction of stricter emission regulations has postponed an increase of engine efficiency. In the following text, the emissions forming due to combustion of Diesel fuel are listed and discussed.

2.2.1. Nitrogen oxides, NO\textsubscript{x}

Nitrogen oxides, NO and NO\textsubscript{2}, commonly referred to as NO\textsubscript{x}, is harmful to the lungs at high concentration, contribute to acid rain and the formation of smog (EPA (1998)). NO\textsubscript{x} is the main local emission from modern Diesel engines. It is primarily formed when air is subjected to high temperatures (Thermal NO\textsubscript{x}), through the extended Zeldovich mechanism, Heywood (1988). For direct injected Diesel engines, high temperature zones are located in the border between the fuel spray and the compressed air.

2.2.2. Particles

The second major emission from modern diesel engines is particles. Particles are suspected to increase the risk for cancer, and has 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 also reduced the allowed limit of particle matter drastically during the years. In the Euro VI legislations the total number of particles is targeted as well as the total mass, discussed in Section 2.3.

2.2.3. *Unburned hydrocarbons, HC*

Unburned hydrocarbons are formed in regions where the fuel for different reasons does not fully burn. This can be caused by too low temperatures and/or lack of oxygen. Emitted into the atmosphere, HC is a health hazard and can produce smog. High production of HC is generally coupled with poor fuel consumption. Diesel engines generally produce low amounts of HC, Heywood (1988).

2.2.4. *Sulfur dioxide, SO$_2$*

Fossil fuels contain different amounts of sulfur, but most of the sulfur is removed 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, which mixed with water will create sulfuric acid leading to acid rain, Heywood (1988). Sulfur dioxide can also form sulfuric acid which might harm the engine as it is highly corrosive. Reduction of SO$_2$ emissions is mainly achieved by reducing the sulfur content in the fuel.

2.2.5. *Carbon monoxide, CO*

Carbon monoxide, CO, is an intermediate step in the combustion between HC and Carbon dioxide. The oxidation of CO to CO$_2$ mainly occurs with the help of OH-radicals. This process requires oxygen and temperatures above 1200K. Interruption of the process can be caused by local lack of oxygen or low temperature, Heywood (1988). 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).

2.2.6. *Water*

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

2.2.7. *Carbon dioxide, CO$_2$*

Carbon dioxide is a byproduct in all combustion process including hydrocarbons. Carbon dioxide is inert, not harmful to animals and vital to plants. However, Carbon dioxide is known to be a greenhouse gas, see Arrhenius (1896). It is generally accepted that it has a non-negligible impact on the global climate, see IPCC (2007).
2.3. Emission Legislation

In Europe, heavy-duty vehicles must conform to the European emission regulation framework, commonly named Euro followed by a number. The first regulation framework was introduced in 1992 and called Euro I. The allowed level of pollutants has since been significantly reduced, see Figure 2.1. Next year, 2014, all new engines in the European Union must meet the Euro VI regulations.

![Figure 2.1. European emission legislation normalized with Euro I emission level, Heavy duty Diesel. data: Dieselnet (2013)](image-url)

2.3.1. Coming legislation

As a direct cause of the introduction of stricter emission legislation, the improvements in combustion efficiency have been postponed due to the relation to combustion temperature. Temperature is namely important in all combustion related chemical processes. In order to obtain maximum fuel efficiency, and low CO₂ emissions, all available fuel should combust at TDC (preferable isentropically). However, if obtained, this would lead to temperatures where formation of NOₓ is very rapid. There is thus a compromise between maximum fuel efficiency and NOₓ emissions. Between the introduction of the Euro I and Euro V legislation (1992-2009) the global emissions of carbon dioxide increased...
with 43 %, see Figure 2.2. Therefore, reduced carbon dioxide emissions are likely to be implemented in future legislation, Euro VII+.

![CO₂ emissions graph](image)

**Figure 2.2.** World carbon dioxide emission with lines for easier comparison, data: World Bank (2013)

### 2.4. The Combustion Engine; History, Energy and the Economy

All economic activity requires energy. This is obvious, but an often overlooked fact important to bear in mind when analyzing the role of the combustion engines in society. In this context, the industrial revolution and the following economic growth, essentially non-existing during previous centuries, can be seen as a drastic increase in available energy. In Figure 2.3 it can be observed that the invention of the combustion engine spurred a drastic increase in economic growth, fueled on stored hydrocarbons; coal and later oil. The increase in energy consumption and composition is depicted in Figure 2.4. The coupling between the engine and the economy is that the engine creates wealth by converting fossil fuels into work. Additionally, the work performed by the engine is much greater than the work needed to extract the fuel. The work produced by the combustion engine led to a drastic increase in living standard and more than two hundred years of exceptional growth. This coupling between available energy and economic growth was understood by early economists, Smith (1776), with land being the primary source of energy through photosynthesis. Due to the fall in the cost of energy with the industrial revolution other factors limited economic growth. Therefore, the coupling between the economy and land/energy was lost in newer economics theories such as Keynesian economics,
Keynes (1936), as other factors became limiting. The belief between a decoupling of land/energy and the economy is well stated by the libertarian think tank Adam Smith Institute, Butler (2011):

"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 to economic growth."

But as conventional crude (cheap) oil production peaked in 2006, IEA (2010), increase in oil production has been forced to rely on unconventional oil such as shale oil. However, this transition has led to almost perfectly inelastic oil supply as reported by Murray & King (2012), i.e. an increase in oil price do not lead to an increase of supply, see Figure 2.5. Therefore, continuing increase in fuel (oil) price is likely. Kumhof et al. (2012) estimates a near doubling (from $100/barrel) of the oil price in the coming decade, yet arguing that their model is not pessimistic from a geological view. Moreover, unconventional oil is generally greater in density and with higher sulfur content and is therefore not as suitable as engine fuel as conventional light sweet crude. From this perspective, it is likely that fuel economy, and diversity, will be an even higher priority in the coming decade. Increasing the demand of more efficient engines.

Figure 2.3. 2000 years of gross world production growth with some historical landmarks, data: DeLong (2000) and for 2000-2011 World Bank (2013)
2.4. THE COMBUSTION ENGINE; HISTORY, ENERGY AND THE ECONOMY

Figure 2.4. World Energy consumption, source: BP (2012) and Grübler (2003)

Figure 2.5. Inflation adjusted oil price as a function of oil supply with colors representing different time eras, data from BP (2012). The leftmost spike being due to the formation of OPEC in 1973 and due to the wake of the Iranian revolution in 1979.
Flow of Internal Combustion Engines

The flow structures found in 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 introduce angular momentum around the center of gravity as well as small-scale turbulence, where the angular momentum forms coherent structures such as Swirl and Tumble, see Section 3.1. However, 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. (2001).

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 the swirl and the turbulence level, Section 3.1.

If the piston and the combustion chamber have been designed in order to create squish, an inward fluid motion will be introduced at the end of the compression stroke, see Section 3.2.

3.1. Swirl and Tumble

Engine swirl number, SN, is defined as the air rotational speed around the cylinder vertical center axis, $\Omega_{\text{Swirl}}$, divided by the rotational speed of the crank shaft, $\Omega_E$, Eqn. (3.1). Tumble number is defined as the air rotational speed around an axis perpendicular to swirl passing the cylinder center of gravity, Eqn. (3.2).

$$SN = \frac{\Omega_{\text{Swirl}}}{\Omega_E} \quad (3.1)$$

$$TN = \frac{\Omega_{\text{Tumble}}}{\Omega_E} \quad (3.2)$$
Flow angular velocity \((\Omega_{\text{Swirl}}, \Omega_{\text{Tumble}})\) is calculated as:

\[
\vec{\Omega} = I^{-1} \vec{L}
\]  

(3.3)

where \(\vec{L}\) is the angular momentum around the center of gravity, evaluated for a finite, \(N\), number of cells as:

\[
\vec{L} = \sum_{k=1}^{N} \vec{r}_k \times \vec{U}_k \rho_k \Delta V_k
\]  

(3.4)

and \(I\) is the moment of inertia:

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

(3.5)

where \(\vec{r}\) is the distance from the center of gravity and \(\mathbf{E}\) is the identity tensor. Angular velocity around the swirl (cylinder) axis is calculated as:

\[
\Omega_{\text{Swirl}} = \vec{\Omega} \cdot \vec{e}_z
\]  

(3.6)

where \(\vec{e}_z\) is the unit vector in the cylinder axis

\[
\Omega_{\text{Tumble}} = |\vec{\Omega} - \Omega_{\text{Swirl}} \cdot \vec{e}_z|
\]  

(3.7)

The swirl and tumble motions are created during intake by the direction of the intake ports. If a swirling motion is wanted, the intake ports will be directed in the tangential direction of the cylinder. Thus, the size of the different motions can be chosen by careful intake channel design.

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 center axis is conserved (if viscosity losses are small), the evolution of swirl depends solely on moment of inertia around this axis. For a piston with bowl, the moment of inertia is decreased (mass is directed inward) and swirl is increased during the late part of compression. If the piston and cylinder head is flat, swirl angular momentum will decrease slightly due to the viscosity losses whereas the moment of inertia will remain un-affected. The tumble motion on the other hand is affected by both a change in angular momentum (will be discussed in next section) and a decrease in moment of inertia. The decrease of moment of inertia will act in order to increase the tumble. The large-scale structures found in IC-engines are used in different manners; the purpose of the swirling flow, primary used in Diesel engines, is to reduce soot caused by wall interaction with the fuel spray. Swirl deflects the fuel spray, increasing the apparent distance between the fuel nozzle and the piston wall. Tumbling flow, on the other hand, is mainly used in spark ignition engines to increase the air-fuel mixing caused by the breakdown of the large-scale structures into small-scale turbulent motions, Lumley (2001).
3. FLOW OF INTERNAL COMBUSTION ENGINES

Swirl number is measured under steady conditions in a so called swirl test rig. The measurements are done for different valve lifts using a honeycomb torque meter, Tippelmann (1977), or a turbine flow meter, Thien (1964). A constant axial velocity profile is assumed for the honeycomb measurements. For the paddle a solid body type rotation is assumed additionally to constant axial velocity. Tumble number can be estimated by tilting the cylinder head ninety degrees and using the test rig. A characteristic swirl and tumble number is then 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. Hence, similar numbers can be obtained by qualitatively different flows. Generally, the measured swirl number is an overestimation as compared to the actual swirl number, Crnojevic et al. (1999).

3.1.1. 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) and Fogleman et al. (2004). 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, showed that a solid body type of rotation is unstable around the second of inertia, arguing that this may be applied to a tumbling flow. However, Gledzer & Ponomarev (1992) found that instabilities also occur around the major, assumed stable, axis. According to Lundgren & Mansour (1995), conversion of an elliptic vortex with aspect ratio of two 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.

Moreover, it has been observed that when injecting air into a tumbling flow, breakdown is amplified (independent of injection direction). Thus, it can be hypothesized that greater turbulence levels lead to faster breakdown of the tumbling motion, Moreau et al. (2005). While tumble is being converted into small scale turbulence around 50 degrees before TDC, see Fogleman et al. (2004), dissipation occurs at TDC ±10CAD, cf. Buschbeck et al. (2012).

3.2. Squish flow

If the piston has a bowl, this will force the flow into the piston bowl at the end of compression, see Figure 3.1. 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 in moment of inertia.
3.3. In-cylinder Turbulence

For in-cylinder flow, there are different methods available to calculate a mean velocity. However, no clear definition of in-cylinder turbulence exists, as turbulence is usually measured as the deviation from the mean the observed value will be highly influenced by the averaging procedure. The most commonly used averaging procedures are: time-averaging, spatial-averaging and phase-averaging.

3.3.1. 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 valve and cylinder. 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. RANS simulations is described in Section 4.7.1.

3.3.2. Spatial-Averaging

For homogeneous turbulence, a spatial average is the most appropriate average. Concerning in-cylinder turbulence, it is not homogeneous. However, it can be argued that the smallest turbulence scales are independent of the larger scales. Therefore, modeling the smallest scales as carried out for Large-eddy
3. FLOW OF INTERNAL COMBUSTION ENGINES

Simulations is a reasonable approach, see Section 4.7.2. However, the turbulence levels will if a spatial average is used depend on the spatial domain.

3.3.3. 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.

3.3.4. Additional method

In this work, an additional 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 and are of solid body rotating type, are described as unstructured. This method and principle thereof will be described in more detail in Section 4.3.
CHAPTER 4

Models and Methods

In Section 4.1 tensor notation and Einstein summation convention is used. 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, which can be written in the following form;

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

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

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

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

\[ S_{ij} = \frac{1}{2} (\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) \]

In order to close the equations an equation of state is needed. It is provided by the ideal gas law;

\[ p = \rho RT \]

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 is used:

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

(4.7)

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

4.2. Vorticity

Studying the vorticity and vorticity equation during compression can provide additional understanding of the 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 (1971): "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:

\[
\vec{\omega} = \vec{\nabla} \times \vec{u}
\]

(4.8)

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

\[
\frac{\partial \vec{\omega}}{\partial t} + (\vec{u} \cdot \vec{\nabla}) \vec{\omega} = (\vec{\omega} \cdot \vec{\nabla}) \vec{u} - \frac{1}{\rho^2} \vec{\nabla} \rho \times \vec{\nabla} p + \vec{\nabla} \times \left( \frac{\vec{\nabla} \cdot \tau}{\rho} \right)
\]

(4.9)

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:

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

(4.10)

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, true for heavy duty engines, the change in cell volume and density is equal the change in total cylinder volume and mean density.
4.3. Kinetic energy

In-cylinder flow velocity can be assumed to be composed of three parts; $\vec{V}_\Omega$ being velocity introduced by a solid body type rotation, the velocity introduced by the piston, $U_p\vec{e}_z$, where $\vec{e}_z$ is the unit vector in the cylinder axis, and the unstructured flow such as turbulence, $\vec{U}_t$. From this viewpoint, the total flow kinetic energy, $E$, can be decomposed into terms consisting of the kinetic energy arising from these different velocities. Firstly, the flow is decomposed into the three velocity components;

$$\vec{U}_k = \vec{U}_{\Omega,k} + U_p,k\vec{e}_z + \vec{U}_{t,k}$$  \hspace{1cm} (4.11)

where subscript $k$ is cell index, the subscript $\Omega$ indicates velocity of a solid body type rotation, $p$ is due to the piston speed and $t$ indicates unstructured motions. The total kinetic energy, neglecting unresolved scales, is thereafter calculated as;

$$E = \frac{1}{2} \sum_{k=1}^{N} \vec{U}_k^2 \rho_k \Delta V_k$$  \hspace{1cm} (4.12)

where $N$ is the number of cells $\vec{U}$ is the velocity vector, $\rho$ is the density and $\Delta V$ is the cell volume. Inserting Eqn. (4.11) into Eqn. (4.12), the following is obtained;

$$E = \frac{1}{2} \sum_{k=1}^{N} (\vec{U}_{\Omega,k}^2 + U^2_{p,k} + \vec{U}_{t,k}^2 + 2\vec{U}_{\Omega,k} \cdot \vec{U}_t \cdot k + 2U_p \cdot k \vec{e}_z \cdot \vec{U}_t \cdot k) \rho_k \Delta V_k$$  \hspace{1cm} (4.13)

rewritten as;

$$E = \frac{1}{2} \sum_{k=1}^{N} (\vec{U}_{\Omega,k}^2 + U^2_{p,k} + \vec{U}_{t,k}^2 + 2U_{\Omega,k} U_{p,k} \vec{e}_z \cdot \vec{U}_t \cdot k + 2U_p \cdot k \vec{e}_z \cdot \vec{U}_t \cdot k) \rho_k \Delta V_k$$  \hspace{1cm} (4.14)

The kinetic energy, consisting of four components, can now be written as;

$$E_{\Omega} + E_p + E_{\Omega p} + e$$  \hspace{1cm} (4.15)

$$e = \frac{1}{2} \sum_{k=1}^{N} (\vec{U}_{\Omega,k}^2 + 2U_{\Omega,k} \vec{U}_t \cdot k + 2U_p \cdot k \vec{e}_z \cdot \vec{U}_t \cdot k) \rho_k \Delta V_k$$  \hspace{1cm} (4.16)

Considering the first contribution to the kinetic energy as written above, the solid body type rotation is defined as;

$$\vec{U}_\Omega = \vec{r} \times \vec{\Omega}$$  \hspace{1cm} (4.17)

where $\vec{r}$ is the distance from the center of gravity and $\vec{\Omega}$ is the angular velocity of the solid body motion, Eqn. (3.3). Thus, the corresponding kinetic energy
due to the solid body rotation is:

\[ E_\Omega = \frac{1}{2} \dot{\Omega}^T I \dot{\Omega} \]  \hspace{1cm} (4.18)

where \( I \) is the moment of inertia around the center of gravity calculated using Eqn. (3.5).

The second contribution is from the piston, which can be assumed to introduce a flow movement equal to the piston speed, \( U_{pist} \), by the piston and linearly decreasing to zero at the cylinder head, Eqn. (4.19).

\[ U_p = U_{pist} \frac{z}{z_p} \]  \hspace{1cm} (4.19)

where \( U_{pist} \) is the piston speed, \( z \) is the distance from the cylinder head, \( z_p \) is the distance between cylinder head and piston. The velocity induced close to the piston and cylinder head is easily motivated and the linear decrease is motivated by the low, compared to the speed of sound, velocity of the piston. The energy connected to the piston motion can consequently be calculated as:

\[ E_p = \frac{1}{2} \int_0^{z_p} \left( U_{pist} \frac{z}{z_p} \right)^2 \rho A dz = 0.5 U_{pist}^2 \frac{m}{3} \]  \hspace{1cm} (4.20)

where \( A \) is the cylinder area and \( m \) is the in-cylinder fluid mass.

The correlating contribution between piston motion and solid body rotation, \( E_{\Omega p} \), is zero due to symmetry conditions.

The last component is the remainder of the kinetic energy and is assumed to be mostly turbulence. This is correct if the mean flow consists of a pure solid body motion with no other coherent motions. For the studied flow this is a valid approximation.

For normalization purposes, the energy of the mean piston speed is calculated as:

\[ MPEnergy = 0.5 m U_{mp}^2 \]  \hspace{1cm} (4.21)

where \( U_{mp} \) is the mean piston speed.

4.4. Turbulent anisotropy

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

\[ b = \frac{< \bar{u} \bar{u}^* >}{q^2} - \frac{1}{3} E \]  \hspace{1cm} (4.22)

and its scalar invariants:

\[ II = b : b \]  \hspace{1cm} (4.23)

\[ II = (bb) : b \]  \hspace{1cm} (4.24)

where : indicates the double dot product, \( \bar{u} \) is the fluctuating velocity vector, \( E \) is the identity tensor and \( q^2 \) is the trace of the Reynolds stress tensor, \( q^2 = < \).
4.5. PROPER ORTHOGONAL DECOMPOSITION

\[ \vec{u}' \cdot \vec{u}' >0 \]
It can be concluded that \( b \) has zero trace and vanishes, together with \( II \) and \( III \), for isotropic turbulence. Additionally, each component in \( b \) 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:

\[
II_{\text{axi}} = \frac{3}{2} \left( \frac{4}{3} |III| \right)^{2/3} \tag{4.25}
\]
and two component turbulence:

\[
II_2 = \frac{2}{9} + 2III \tag{4.26}
\]

If these boundaries are plotted one obtain what is now known as the Lumley triangle, Figure 4.1.

\[ \text{Figure 4.1. Anisotropy-invariant map (Lumley triangle)} \]

4.5. Proper Orthogonal Decomposition

4.5.1. Background

Proper orthogonal decomposition (POD) was first proposed to fluid flow by Lumley (1967). The aim was to detect the energetic coherent structures in the flow that characterize the fluid flow. In POD, the flow field is decomposed into
4. MODELS AND METHODS

ortho-normal modes ordered after energy content. Coherent structures are the most energetic structures in the flow field and will be represented by the first modes. For a statistically stationary flow, the zeroth, i.e. most energetic, mode will correspond to the mean flow field. The fluctuating energy modes can then be divided into coherent structures with high energy content and an incoherent part having low energy content.

\[
\vec{u}(\vec{x}, t) = \vec{\phi}_0(\vec{x}) + \sum_{j=1}^{J} a_j(t) \vec{\phi}_j(\vec{x}) + \sum_{j=J+1}^{\infty} a_j(t) \vec{\phi}_j(\vec{x}),
\]

where \( j \) denotes the node number, \( a_j(t) \) the corresponding time coefficient, \( \vec{\phi}_j \) the POD mode and \( \vec{u}(\vec{x}, t) \) is the known flow field at time \( t \). However, choosing \( J \) is subjective and different values can be appropriate for different flow phenomena. POD therefore provides the possibility to make coherent structures more visible by removing incoherent motions.

4.5.2. POD algorithm

POD is decomposed such that the energy content should be as large as possible in the first mode and decreasing with increasing mode number. Mathematically, this is formulated as;

\[
\vec{\phi} \in \text{max}_{L^2(0,1)} \left\{ \frac{|\langle \vec{u}(\vec{x}, t), \vec{\phi}(\vec{x}) \rangle|^2}{\| \vec{\phi}(\vec{x}) \|^2} \right\}
\]

where \( \langle \vec{u}, \vec{\phi} \rangle \) is the inner product, \( \| \cdot \| \) is the \( L^2 \) norm, and \( \langle \cdot \rangle \) denotes the time average.

In practice, data is available in the form of snapshots, taken at regular intervals, with a discrete number of sampling points. In the following section, it is assumed that the data containing the velocity field, \( \vec{u} = (u, v, w) \), used for POD is in the form of \( M \) snapshots and that the number of grid points is \( N \).
To compute POD, the data is collected in the matrix $X$,

$$
X = \begin{bmatrix}
    u(\vec{x}_1, t_1) & \cdots & u(\vec{x}_1, t_M) \\
    \vdots & \ddots & \vdots \\
    u(\vec{x}_N, t_1) & \cdots & u(\vec{x}_N, t_M) \\
    v(\vec{x}_1, t_1) & \cdots & v(\vec{x}_1, t_M) \\
    \vdots & \ddots & \vdots \\
    v(\vec{x}_N, t_1) & \cdots & v(\vec{x}_N, t_M) \\
    w(\vec{x}_1, t_1) & \cdots & w(\vec{x}_1, t_M) \\
    \vdots & \ddots & \vdots \\
    w(\vec{x}_N, t_1) & \cdots & w(\vec{x}_N, t_M)
\end{bmatrix}
$$

This matrix is of the size $3N$ by $M$. To calculate the POD modes, the method of Lagrange multipliers is used to reform the maximization problem, Eqn. (4.28), into an eigen-value problem, see e.g. Holmes et al. (1998). On a discretized, form the eigen-value problem takes the following form;

$$
XX^T \vec{\phi}_j = \vec{\lambda}_j \vec{\phi}_j
$$

where $\vec{\lambda}_j$ corresponds to the kinetic energy of the mode $j$. Since $N$ is generally large, special treatment is necessary in order to find these eigen-values. To address this the so called method of snapshots is used which reduces the computational cost significantly, Sirovich (1987).

4.6. Particle Image Velocimetry

Parts of the results presented in this thesis were obtained using Particle Image Velocimetry (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).

4.6.1. The basics

The basic principle behind PIV is rather simple; seed particles are introduced into the flow, from two subsequent images, the distance traveled by the particles between the two snapshots are measured. 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).

4.6.2. 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. In the work presented in this thesis black tape was used to stop reflections to hit the camera. However, this led to that not all of the entire area of the cylinder could be measured.

4.7. Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is used in order to calculate fluid flow. 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 industrial applications. Thus, modeling is 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 (LES) attractive for more complex geometries.

4.7.1. Reynolds Averaged Navier-Stokes, RANS

The RANS equations are obtained by decomposing the flow into a mean and a fluctuating component and thereafter 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 some model equations has to be applied. The most common "all-scale" turbulence model is the two equation $k$-$\varepsilon$ RANS model. Although widely used, the $k$-$\varepsilon$ model has some 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 are some examples. 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 isentropic eddy viscosity assumption will fail in predicting the physics correctly. However, from an engineering point they are very useful as they often give qualitatively useful results.

4.7.2. LES

As a contrary to RANS simulations where all turbulent structures are modeled, Large Eddy Simulations only model 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).

In this work, the smallest scales have been 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. This can be done as the equilibrium subrange is independent on viscosity. Furthermore, several studies show that this methodology provides accurate and in some particular cases better results as compared to explicit Sub-Grid Scale (SGS) models, such as the Smagorinsky-Lilly model, see e.g. Patnaik et al. (2003); Aspden et al. (2008) and Fernando F. Grinstein & Rider (2007) or (Pope 2008, p. 631-634). For compressing flows, 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, Section 4.2. Thus, the turbulence cannot be assumed isotropic even at the smallest scales. For a flow with increasing enstrophy $\frac{1}{2} (\vec{\omega} \cdot \vec{\omega})$ it can be observed that the energy spectra do not follow the $-5/3$ "law", cf. Orlandi & Pirozzoli (2009). 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.

4.7.3. Numerical set-up

This section will focus on the numerical set-up used in this work.
4. MODELS AND METHODS

4.7.3a. Discretization/Numerical schemes. In fluid dynamics, the convection diffusion equation is of uttermost importance, and can be written as:

\[
\frac{\partial \phi}{\partial t} + \vec{\nabla} \cdot (\vec{u} \phi) = \vec{\nabla} \cdot (D \vec{\nabla} \phi) \quad (4.31)
\]

where \( \phi \) is the variable of interest, \( \vec{u} \) the velocity and \( D \) is the diffusivity. If Eqn. (4.31) is integrated over a control volume, \( V \), and Gauss’ theorem is applied Eqn. (4.32) is obtained.

\[
\int_V \frac{\partial \phi}{\partial t} dV + \int_S (\vec{u} \phi) \cdot \vec{n} dS = \int_S (D \vec{\nabla} \phi \cdot \vec{n} dS \quad (4.32)
\]

where \( S \) is the surface of the of the control volume and \( \vec{n} \) is the surface normal vector. Furthermore, if the volume is discretized with a finite number of faces the following is obtained:

\[
\int_S (\vec{u} \phi) \cdot \vec{n} dS \approx \sum \vec{u}_c \cdot \vec{n}_c \partial S_c \quad (4.33)
\]

and

\[
\int_S (\rho D \vec{\nabla} \phi) \cdot \vec{n} dS \approx \sum \left( \rho D \vec{\nabla} \phi_c \right) \cdot \vec{n}_c \partial S_c \quad (4.34)
\]

where the index \( c \) denotes values on the middle of the surfaces. In order to solve these equations the advective, \( \phi_c \), and diffusive, \( (\vec{\nabla} \phi_c) \), fluxes need to be approximated. Generally the diffusive fluxes, \((\vec{\nabla} \phi_c)\), is approximated using by 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.

The approximation of \( \phi_c \) is illustrated for the one-dimensional case in Figure 4.2. A simple method to approximate \( \phi_c \) would be a linear approximation;

\[
\phi_c \approx \phi_P + \frac{\phi_E - \phi_P}{x_E - x_P} \delta x_{Pc} \quad (4.35)
\]

**Figure 4.2.** Control volume with designations and flow direction
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;

\[ \phi_e \approx \phi_P \] (4.36)

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

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

\[ \phi_e \approx \phi_P + \frac{1}{2} \psi(\phi_E - \phi_P) \] (4.37)

where \( \psi \) is a function of

\[ r = \frac{\phi_P - \phi_W}{\phi_E - \phi_P} \] (4.38)

or the ratio between upwind-side gradient and 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) \( \psi(r) = \frac{3 + r}{4} \)

4.7.3b. Total Variation Decreasing (TVD). In order for a scheme to be TVD is that local minima must be non-decreasing and local maxima must be non-increasing, i.e. it preserves monotonicity. For monotonicity to be satisfied, the Total Variation (TV) must not increase, see Eqn. (4.39) and Figure 4.3, hence the name.

\[ TV = |\phi_2 - \phi_1| + |\phi_3 - \phi_2| + |\phi_4 - \phi_3| + |\phi_5 - \phi_4| \] (4.39)

**Figure 4.3.** An example of a discrete data set for illustrating total variation

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 $\psi(r) \leq 2$

Hence 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 $\frac{3}{4} \leq r \leq 5$

This implies that in order to make a scheme TVD, the range of possible values of the additional convective flux, $u_i\psi(r)\frac{\phi_E-\phi_P}{2}$, must be limited. Hence, $\psi(r)$ 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, Figure 4.4. Region of second order accurate TVD schemes can be seen in Figure 4.4. In this work the limiter function introduced by Sweby (1984) has been used, Eqn. (4.40), with $\beta = 1$.

![Figure 4.4. Region for second order accurate TVD scheme](image)

$$\psi(r) = \max[0, \min(\beta r, 1), \min(r, \beta)] \quad (4.40)$$
CHAPTER 5

Flow under consideration

A brief summary of the work and results presented in Paper 1 and 2, regarding the flow in a steady swirl test rig, is presented in Section 5.1 and 6.1. The effect of compression presented in Paper 3 and 4 is described in Section 5.2.1 and the results are presented in Section 6.2. Conclusion for the intake and compression cases can be found in Section 7.1 and Section 7.2, respectively.

5.1. Steady flow rig

5.1.1. Geometry

LES computations and PIV measurements were carried out on a model of a steady swirl flow rig. For the experiments an impulse torque swirl rig located at Scania were used. It consists of a cylinder head mounted on a transparent glass cylinder with a diameter of 130 mm and a height of 220 mm. The glass cylinder was placed on a cubic box from which air was drawn by a pressure drop controlled pump. Constant valve lift and pressure were used throughout each experiment, i.e. the flow is statistically stationary.

Generally, results presented in Section 6.1 are scaled with the bulk velocity and the nominal swirl number unless otherwise stated. Engine swirl number is defined as the air rotational speed around the cylinder vertical center axis, $\Omega_{\text{Swirl}}$, divided by the rotational speed of the crank shaft, $\Omega_E$, Eqn. (3.1). For a steady swirl test rig the swirl number is calculated using Eqn. (5.1) and (5.2), where $M_z$ is the flux of angular momentum through a plane perpendicular to the cylinder axis, $\rho$ is the density, $\vec{V}$ is the velocity vector, $\vec{r}$ is the distance from the cylinder axis, $\vec{e}_z$ is the unit vector parallel to the cylinder axis, subscript $k$ denotes k:th cell, $\Delta A_k$ is the surface area of the cell $k$, $S$ is the piston stroke, $\bar{\rho}$ is the mean density and $Q_m$ is the measured mass flow. Positive swirl is defined in the counter clockwise direction.

$$M_z = \sum_{k=1}^{N} \rho_k (\vec{V}_k \times \vec{r}_k) \vec{e}_z V_{z,k} \Delta A_k$$ \hspace{1cm} (5.1)

$$SN = \frac{2SM_z \bar{\rho}}{Q_m^2}$$ \hspace{1cm} (5.2)
Table 5.1. Experimental setup

<table>
<thead>
<tr>
<th>Case</th>
<th>Valve lift</th>
<th>Pressure drop</th>
<th>Air density</th>
<th>Room temperature</th>
<th>Bulk velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[mm]</td>
<td>[Pa]</td>
<td>[kg/m³]</td>
<td>[°C]</td>
<td>[m/s]</td>
</tr>
<tr>
<td>A₁</td>
<td>5</td>
<td>800</td>
<td>1.195</td>
<td>21</td>
<td>2.61</td>
</tr>
<tr>
<td>A₂</td>
<td>5</td>
<td>1400</td>
<td>1.193</td>
<td>21</td>
<td>3.43</td>
</tr>
<tr>
<td>A₃</td>
<td>5</td>
<td>2400</td>
<td>1.188</td>
<td>21</td>
<td>4.57</td>
</tr>
<tr>
<td>B₁</td>
<td>11</td>
<td>800</td>
<td>1.195</td>
<td>21</td>
<td>4.02</td>
</tr>
<tr>
<td>B₂</td>
<td>11</td>
<td>1400</td>
<td>1.193</td>
<td>21</td>
<td>5.36</td>
</tr>
<tr>
<td>B₃</td>
<td>11</td>
<td>2400</td>
<td>1.188</td>
<td>21</td>
<td>7.19</td>
</tr>
<tr>
<td>C₁</td>
<td>15</td>
<td>800</td>
<td>1.195</td>
<td>21</td>
<td>4.26</td>
</tr>
<tr>
<td>C₂</td>
<td>15</td>
<td>1400</td>
<td>1.193</td>
<td>21</td>
<td>5.67</td>
</tr>
<tr>
<td>C₃</td>
<td>15</td>
<td>2400</td>
<td>1.188</td>
<td>21</td>
<td>7.59</td>
</tr>
</tbody>
</table>

5.1.2. PIV setup

A stereo PIV setup was used, with the two cameras mounted in a backward-backward configuration as shown in Figure 5.1. 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 particles were used to seed the flow. As displayed in Figure 5.1 there is no inlet channel in which seeding particles 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 the entire area of the cylinder was not measurable, see Figure 5.1. The cases presented in Table 5.1 were measured having a resolution of 0.6 × 0.6 mm. The three chosen valve lifts corresponds to an area opened by the valves (curtain area) to intake port area ratio of 50, 100 and 150 %.

5.1.3. LES setup

For the LES simulations the outlet box was removed instead the cylinder was extruded so that the outlet was positioned more than three diameters downstream of the measurement plane. The measurement plane and computational geometry are shown in Figure 5.1. The Implicit LES methodology described in Section 4.7.2 was used in combination with the TVD scheme described in Section 4.7.3a. Simulations were carried out on cases A₂, B₃ and C₃ in Table 5.1. Mass flow and temperature were set at the inlet and at the outlet, a static pressure, matching the experiments, was applied. In order to resolve over one decade of the inertial subrange a mesh with a specified cell width of 0.7 mm was deemed sufficient, cf. Paper 1. An automatic mesher (FIRE®) was used to produce the meshes.
Figure 5.1. PIV measurement setup. Position and size of the measurement plane, with position of the valves marked. Flow inlet direction indicated by arrows.
5. FLOW UNDER CONSIDERATION

Table 5.2. Engine Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore [mm]</td>
<td>130</td>
</tr>
<tr>
<td>Stroke [mm]</td>
<td>160</td>
</tr>
<tr>
<td>Conrod length [mm]</td>
<td>255</td>
</tr>
<tr>
<td>Clearance height [mm]</td>
<td>9.5</td>
</tr>
<tr>
<td>Engine speed [rpm]</td>
<td>1200</td>
</tr>
<tr>
<td>Engine load [%]</td>
<td>0</td>
</tr>
</tbody>
</table>

5.2. Effects of compression

5.2.1. Swirling and tumbling flow

The effects of compression on a swirling/tumbling flow in a simplified motored engine were studied using Large-Eddy Simulations. The geometry studied was a simple cylinder with an artificially created swirling/tumbling motion. Engine setup can be found in Table 5.2 and a graphical view of the initial condition is displayed in Figure 5.2. Crank angles 180-380 were simulated, where 180° corresponds to Bottom Dead Center and 360° to firing Top Dead Center. Thus, 20 crank angles were simulated after firing TDC.

Figure 5.2. Geometry, and initial flow field at BDC
5.2. EFFECTS OF COMPRESSION

5.2.1a. Change of conrod length. In order to study the effect of piston velocity the length of the conrod was changed to be slightly longer than half the stroke, i.e. 81 mm instead of 255 mm. The change of length will increase the acceleration of the piston and maximum speed significantly. Thus, highlighting the effect of piston speed on flow quantities.

5.2.1b. Change of compression ratio. When changing the conrod length both the piston velocity as well as the relative change of volume will change, see Eqn. 4.10. If the compression ratio is changed instead the speed of the piston is kept constant while the dilatation is changed. However, in order to ensure that the simulations are started having the same initial conditions, the stroke length was decreased 2 mm reducing maximum piston velocity slightly. The combined changes gave a compression ratio of 14.7 as compared to the original 17.8.

5.2.2. Isotropic turbulence in a swirling flow
Solid body type rotation with swirl numbers, Eqn. 3.1, corresponding to 0, -1 and -4 was initialized. Isotropic turbulence with turbulent kinetic energy of 23 m²/s² was superimposed on the swirling motion. Engine parameters presented in Table 5.2 were used.

5.2.3. Numerical setup
An Arbitrary Lagrangian-Eulerian (ALE) mesh motion technique were used, i.e. mesh is compressed with the geometry. In order to avoid cells with to high aspect ratio, which would reduce accuracy significantly see Celik et al. (2001), the results were mapped to a new mesh once the geometry were compressed a factor two. For the compression stroke remeshing had to be done four times for the original compression ratio and three times for the case with a compression ratio of 14.7, i.e. five and four different meshes were used, respectively. 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, cf. Lumley (1999). A convective Courant number below 0.2 were used at all times together with a PISO pressure coupling algorithm, Issa (1986).

5.2.4. Mesh
Meshes were created by setting an apparent mesh cell size, $\Delta x$ [mm], the number of cells for the mesh at TDC in direction A-C was calculated as:

$$A = \frac{2\pi R_0}{6\Delta x}$$  \hspace{1cm} (5.3)

$$B = \frac{2\pi R_0}{9\Delta x}$$  \hspace{1cm} (5.4)
5. FLOW UNDER CONSIDERATION

\[ C = \sqrt{2}\frac{\text{Clearance}[\text{mm}]}{\Delta x} \]  \hspace{1cm} (5.5)

A graphical view of a 3 mm mesh is depicted in Figure 5.2.4

![Figure 5.2.4: A graphical view of a 3 mm mesh](image)

**Figure 5.3.** Example mesh. (3 mm apparent mesh size, stretched position)

The resulting cell number is rounded to closest integer. The length of A was set to 40 mm. Therefore, the cell size in the inner part of the cylinder is slightly smaller than the assumed mesh cell size, \(\Delta x\). Average cell volume for the 0.544 mm mesh was 0.098 mm\(^3\) giving an average cell width of 0.46 mm.
CHAPTER 6

Results and discussion

In this chapter, results regarding flow created during intake is presented in Section 6.1 and results regarding the effect of compression is presented in Section 6.2. Please consult Chapters 2, 3, 4 and 5 for background, methods and case description.

6.1. Results, steady flow rig

All planes shown are depicted from above, and the inlet ports are positioned to create a counter clockwise rotating vortex. LES data has been used to give an introduction to the flow in Section 6.1.1. In Sections 6.1.2 PIV data is used to show how this affects the flow in the PIV measurement plane. Finally, the results are discussed in Section 6.1.3.

6.1.1. Overview

In this section, the flow for the 15 mm valve lift case is presented to give a representation of the different phenomena affecting the flow when entering the cylinder.

The flow enters the domain in the large intake box, it is thereafter accelerated through the intake ports, Figure 6.1. Before the valves, the geometry turns downwards forcing high momentum flow to the opposite side replaced by the low momentum flow close to the walls forming two small vortices of Dean vortex type, Dean & Hurst (1959), which is marked as A1 and A2 in Figure 6.2. However, before reaching the valves the valve stem obstructs the flow creating a wake with a vortex street, B in Figure 6.2. The vortex street is better observed in Figure 6.3 where a plane parallel to the streamlines of the mean flow with in-plane velocity vectors. The flow enters the cylinder as a jet with high velocity gradients and inflection point, giving rise to further instabilities. Therefore, the flow field is very complex with high levels of fluctuations when entering the cylinder. Due to the high complexity simplified numbers such as swirl and tumble have been and are still used extensively within the industry.

A clear difference between the way flow enters the cylinder for the three different valve lifts are shown in Figure 6.4. It is found that a clear vortex street downstream of the valves can be detected for the 11 and 15 mm valve lift cases.
However, for the 5 mm valve lift case the flow is directed in all directions and no recirculation zone is observed behind the valves.

Differences in flux of (axial) angular momentum through different planes can be presented in Table 6.1 for the different valve lifts. It is observed that for the 5 mm valve lift case there is a significant loss of axial angular momentum flux prior to the valves. For the 11 mm valve lift it is observed that the flux of axial angular momentum is increased in the end of the ports, while no change in flux is noticed for the 15 mm valve lift in the domain.

These differences give rise to very different in-cylinder flow fields. Considering the streamlines in the mean flow field, Figure 6.5, it is noticed that for the low 5 mm valve lift two counter rotating vortices are formed while one stable vortex can be found for the 15 mm valve lift case. For 11 mm valve lift one single vortex is formed. However, this vortex differs from the vortex found for the 15 mm valve lift case by being tilted.

Figure 6.1. Flow into the cylinder, 15 mm valve lift
6.1. RESULTS, STEADY FLOW RIG

Figure 6.2. Dean vortices and wake of the valve stem, 15 mm valve lift, left valve

Figure 6.3. Vortex street behind the valve stem, in-plane velocity vectors, left valve, 15 mm valve lift

6.1.2. PIV plane

In the PIV measurement plane, Figure 5.1, it is found that valve lift has a profound influence on the flow structures. No clear vortex is observed in the
6. RESULTS AND DISCUSSION

Figure 6.4. Axial velocity in the end of intake ports, with in cylinder velocity vectors. 5, 11 and 15 mm valve lift, respectively

Table 6.1. Flux of (axial) angular momentum, Eqn. (5.1) (normalized as Eqn. (5.2)) through planes at a distance $\Delta z$ downstream of the cylinder head and a plane crossing the two ports prior to the first turn of the right port in Figure 6.1.

<table>
<thead>
<tr>
<th>$\Delta z$</th>
<th>5 mm</th>
<th>11 mm</th>
<th>15 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ports</td>
<td>1.5</td>
<td>1.2</td>
<td>1.5</td>
</tr>
<tr>
<td>-15 mm</td>
<td>0.2</td>
<td>1.5</td>
<td>1.5</td>
</tr>
<tr>
<td>30 mm</td>
<td>0.2</td>
<td>1.1</td>
<td>1.5</td>
</tr>
<tr>
<td>100 mm</td>
<td>0.2</td>
<td>1.1</td>
<td>1.4</td>
</tr>
</tbody>
</table>

mean flow at 5 mm lift. However, there is indication of two counter rotating vortices. Axial (downward) flow is primarily between the valves, see Figure 6.6. For the higher valve lifts (11, 15 mm), a swirling pattern appears and the axial flow is pressed outwards towards the cylinder walls. Additionally, the swirl number is increasing with increasing valve lift, depicted in Figure 6.7. Changing the driving pressure is observed not to affect either the mean flow field, Figure 6.6, nor in turbulent quantities, Figure 6.9.

Fluctuations in swirl, see Figure 6.7, and turbulence, Figure 6.8, are decreasing with increasing valve lift. For 5 mm valve lift the fluctuations in swirl
is greater than the mean swirl number, while smaller for the higher valve lifts. An indication that the fluctuations in swirl number is of greater importance than mean level at low valve lifts. With increasing valve lift the Reynolds stress tensor is becoming more axisymmetric and isotropic, see Figure 6.9.

The high turbulence level for all cases masks flow structures in the instantaneous flow fields. Decomposing the flow using POD, described in Section 4.5, makes flow structures more visible. In Figure 6.10 the snapshots, reconstructed using POD mode 0-10, with the highest, mean and lowest instantaneous swirl number found in Figure 6.7 is shown together with the mean flow field. For 5 mm lift two vortices with different direction are seen for highest and lowest swirl number snapshot. For the snapshot with a swirl number closest to the mean, i.e. the "Mean" snapshot, the flow is mainly in the axial direction. At 11 mm valve lift no swirling motion is observed for the snapshot with the lowest swirl number while a clear vortex is seen for the snapshot with the highest swirl number. Increasing the valve lift further to 15 mm the difference between the snapshots is mainly in the position of the vortex core.
Figure 6.5. Formation of vortices. Columns represent 5, 11 and 15 mm valve lift, respectively.
Figure 6.6. Scaled mean axial 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.
Figure 6.7. Instantaneous swirl number of the reduced surface, see Figure 5.1. Rows corresponds to 5, 11 and 15 mm valve lift, respectively. Time of snapshots with minimum (A), as close to mean as possible (B) and maximum instantaneous swirl number (C) shown for the 5 mm valve lift.
Figure 6.8. Fluctuations in velocity, reduced surface, see Figure 5.1, with position of the valves marked, 2400 Pa. Rows correspond to fluctuations in axial velocity, tangential velocity and correlation between tangential and axial velocity, respectively. Columns represent 5, 11 and 15 mm valve lift, respectively. Note the difference in scale between 5 mm and the higher valve lifts.
Figure 6.9. Turbulent anisotropy invariant map, colors indicating the distance from the cylinder center in cylinder radius. PIV data, offset of 0.025 on the x-axis between cases. Rows corresponds to 5, 11 and 15 mm valve lift, respectively.
6.1. RESULTS, STEADY FLOW RIG

Figure 6.10. Reconstructed scaled in-plane velocity, POD mode 0-10. 5, 11 and 15 mm valve lift. 2400 Pa pressure drop. For mean flow field and snapshots with, Min, approx. mean and maximum swirl in the four rows, respectively
6.1.3. Discussion

The flow motions of the in the intake port and through the valves is extremely complex. Due to the turning of the intake port secondary motions such as the Dean type vortices shown in Figure 6.2 is initiated. Furthermore, the obstruction of the valve stem to the flow field creates a vortex street, Figure 6.3. As the flow enters the cylinder domain in a jet like manner, further instabilities is introduced. This have a high impact on the flow, observed further downstream in the PIV measurement plane, Figure 5.1. From the conducted experiments and simulations it is found that the valve lift has a major influence on the downstream flow field, Figure 6.6. However, as the reduction of angular momentum flux is prior to the valves, Table 6.1, differences are likely not to be caused by instabilities when the flow enters the. Furthermore, it is observed that for 11 mm valve lift the flux of angular momentum is increased in the end of the ports and subsequently lost over the valves. For the 5 mm valve lift the obstruction caused by the valves are significant enough to hinder axial momentum prior to the valves. It can be seen in Figure 6.4 that the direction of the flow entering the cylinder is highly influenced by the valve lift. The effect of this is clearly seen in the swirl number which is significantly lower for the 5 mm valve lift case than for the higher valve lifts, Figure 6.7.

The difference in the large scale vortex formation noticed for the three valve lifts is found to cause give rise to differences in small scale turbulent strength anisotropy. It is noted that the fluctuations are a magnitude larger for the 5 mm valve lift case than for the higher valve lifts. Additionally, it is also noticed that the turbulent anisotropy differs significantly between the three valve lifts. The turbulent anisotropy noticed for 5 mm valve lift in the PIV measurement plane as well as close to the cylinder head is of interest since it has been shown that small scale turbulence may have a destabilizing effect on tumble, as discussed by Moreau et al. (2005).

For the 5 mm valve lift case it is also found that the fluctuations are larger than the mean number. This is an indication that the mean swirl number is at best an irrelevant flow characteristic for low valve lifts and possible misleading.
6.2. Results, Effects of compression

6.2.1. Swirling and tumbling flow

Initial flow field (180 CAD) is a well ordered swirling motion, see Figure 6.11, displaying low levels of vorticity, Figure 6.12. Most of the kinetic energy of the flow is situated in the swirling motion, $E_\Omega$, Figure 6.13. During compression, the flow becomes more erratic and at 340 CAD, the flow is turbulent, both qualitatively and quantitatively, Figures 6.11 and 6.13. Vorticity levels has also increased, Figure 6.12 as well as the kinetic energy of the unstructured flow, $e$, Figure 6.13. Thereafter, the kinetic energy of the unstructured motions starts to decrease to the end of the simulation (CAD 380), Figure 6.13. Therefore, a more structured swirling motion is observed at the end of compression, Figure 6.11.

During compression vorticity in the cylinder ($z$-) axis is relatively unaffected, Figure 6.14. However, vorticity perpendicular to the cylinder axis is highly affected by the compression. The vorticity magnitude reaches a maximum around the point of maximum dilatation, Figure 6.15.
Figure 6.11. Normalized velocity magnitude (with subtracted velocity induced by the piston) with in-plane vectors. Crank angles, from left to right starting at top-left, 180, 250, 296, 340, 360, 380
6.2. RESULTS, EFFECTS OF COMPRESSION

Figure 6.12. Mid-plane vorticity magnitude. Crank angles, from left to right starting at top-left, 180, 250, 296, 340, 360, 380

Figure 6.13. Kinetic Energy (KE) of the flow, decomposed into; total KW (E), rotational KE (E_Ω), piston induced KE (E_{pist}) and KE of the unstructured motion (e)
6. RESULTS AND DISCUSSION

Figure 6.14. Volume averaged vorticity; around x, y and z-axis

Figure 6.15. Normalized relative volume change (dilatation),
- - mid-stroke, - cylinder height=radius (AR2), - maximum dilatation, - TDC
6.2. RESULTS, EFFECTS OF COMPRESSION

6.2.1a. Effect of changing conrod length. Kinetic energy introduced by the piston is redirected into unstructured fluid motion. By changing the conrod length the maximum piston velocity and dilatation is increased. If the conrod length is set to approximately the crank radius, the piston velocity and acceleration is increased significantly. In Figure 6.16 the kinetic energy is plotted for an engine with a 0.081 m conrod. A comparison between the dilatation for the two different conrod lengths are shown in Figure 6.17. It is observed that the kinetic energy introduced by the piston is increased substantially, Figure 6.16. However, the increase in unstructured kinetic energy is significantly lower.

![Figure 6.16. Kinetic Energy (KE) of the flow. decomposed into; total KW (E), rotational KE (E_Ω), piston induced KE (E_pist) and KE of the unstructured motion (e). Blue color indicating the case with changed conrod length](image)

![Figure 6.17. Normalized relative volume change (dilatation), with maximum position, - TDC. Blue color indicating the case with changed conrod length](image)
6.2.1b. *Effect of compression ratio.* By changing the compression ratio it is found that there is no significant change in unstructured kinetic energy, Figure 6.18, even if the maximum dilatation has been reduced considerably. The results indicate that the kinetic energy introduced by the piston is of greater importance than the absolute value of dilatation.

**Figure 6.18.** Kinetic Energy (KE) of the flow, decomposed into; total KW (E), rotational KE (E_Ω), piston induced KE (E_{pist}) and KE of the unstructured motion (ε). Blue color indicating the case with changed compression ratio.

**Figure 6.19.** Normalized relative volume change (dilatation), - - mid-stroke, - cylinder height=radius (AR2), - maximum dilatation, - TDC. Blue color indicating the case with changed compression ratio.
6.2. RESULTS, EFFECTS OF COMPRESSION

6.2.2. Isotropic turbulence in a swirling flow

The evolution of the flow velocity magnitude during compression can be seen in Figure 6.2.2. It is observed that the initial high turbulent levels dissipate quickly and the swirl is becoming visible. The dissipation is also shown in Figure 6.2.2. However, around 280 CAD, it is noticed that fluctuations in the tangential velocity component start to increase, followed by an increase in fluctuations in the other components. In Figure 6.2.2 it can be seen that the radial length scale of the tangential fluctuations is increasing until 320 CAD when the fluctuations start to decrease. At 320 CAD, the fluctuations in the z- and radial velocity component start to increase. If in-cylinder vortices are visualized using the Q-criteria it is found that the vortices have aligned with the mean flow by 340 CAD.

6.2.2a. Effect of swirl number. An increase in swirl number is found to increase dissipation of the turbulent fluctuations, Figure 6.24. The tangential component is most strongly affected and contrary to the low and zero swirl cases, Figure 6.2.2a.

6.2.3. Discussion

Generally, 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 Boree et al. (2002) among others. However, for the engine studied in the present work the increase in energy can only be originating from the piston as the energy available in the tumble motion is small, for the swirling and tumbling case, and non-existent for the swirl case. What has been noticed in all simulations is an increase in 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. (4.9), giving:

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

(6.1)

Where the direct effects of compression is encircled. Hence, it can be seen that compression only increases vorticity perpendicular to the cylinder axis, increasing turbulent anisotropy. This amplification of vorticity is present even if no global elliptic instability can be found. However, in order to increase the kinetic energy of the unstructured motion energy has to be added to the
6. RESULTS AND DISCUSSION

Figure 6.20. 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, 340, 350, 360, 380.

flow. From the results in Section 6.2.1a and 6.2.1b this is linked to the piston velocity.

The cause of increased tangential fluctuations for the high swirl case, seen in Figure 6.2.2a is likely to be caused by velocity deficit close to cylinder head and piston. However, this needs further investigation. The increase in turbulent length scale noticed in the beginning of the compression in Figure 6.2.2 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,
6.2. RESULTS, EFFECTS OF COMPRESSION

Figure 6.21. Turbulent fluctuations, \((-\,--\,) < u_\theta u_\theta >, \, (-\,-\,-\,-\,) < u_r u_r >, \, (-\,-\,-\,-\,-\,-\,) < u_z u_z >\). Swirl number 1

Figure 6.22. Radial length scale in the mid-plane for different crank angles. Swirl number 1

Figure 6.23. Visualization of vortices using the Q-criteria. Columns corresponds to 180 and 340 CAD, respectively

i.e. an increase in production, more kinetic energy is diffused to the smaller scales of the flow. For the swirling/tumbling flow no such reduction in kinetic
6. RESULTS AND DISCUSSION

**Figure 6.24.** Turbulent intensity, swirl number 0, 1 and 4

**Figure 6.25.** Turbulent fluctuations, $(-) < u'_{\theta} u'_{\theta} >$, $(-\cdot) < u'_{r} u'_{r} >$, $(-\cdot\cdot) < u'_{z} u'_{z} >$. Rows corresponds to swirl number 0 and 4, respectively.

Energy of the unstructured motions are present, why the qualitative reduction of length scale noticed is present.
CHAPTER 7

Summary and conclusions

7.1. Conclusions, steady flow rig

PIV measurements and LES computations were executed on a steady swirl test rig geometry. Three different valve lifts and driving pressures were used in the measurements. During the intake, flow is given direction and thus momentum while flowing through the intake ports. In the end of the ports the flow is directed downward pass the intake valves. Depending on the valve lift different amounts of angular momentum is conserved. Hence, it is found that higher valve lift increases swirl level and decreases turbulence. Turbulence levels after an intake valve is found to be very high, hiding structures in the instantaneous flow field effectively. Fluctuations in swirl number at low valve lifts are found to be greater than the averaged swirl number. It has thus been found that it is insufficient to discuss on averaged swirl number for low valve lifts. The results can be summarized as follows;

- Driving pressure has no effect on neither mean flow field nor turbulent quantities.
- At low valve lifts characteristic swirl number measured in a steady swirl test rig is insufficient to characterize the engine.
- With increasing valve lifts the swirl number increases and the fluctuations in the swirl number decreases.
- Turbulence is found to be more anisotropic at low valve lifts.

7.2. Conclusions, Effects of compression

7.2.1. Swirling and tumbling flow

Large-eddy simulations were performed on a simplified engine geometry. An initialized flow field was compressed and the effects were studied. It was found that the piston provided the kinetic energy of the unstructured motions found at TDC. The unstructured (turbulent) kinetic energy is greatest at the time of maximum dilatation. During early expansion rapid dissipation is found.

From the simulations where conrod length and compression ratio were changed it was noted that kinetic energy introduced by the piston is of greater importance than a change in maximum dilatation. The cause of this may be in the fact that the difference in dilatation was relatively small when piston velocity
was high. However, it can be conclude that an increase in piston velocity will lead to higher turbulence levels.

7.2.2. Isotropic turbulence in a swirling flow

By studying the effect of compression on swirling motions of different strengths it was found that an increase in swirl number will suppress initial turbulence. The transverse length scale of the fluctuations in tangential velocity is affected by compression by increasing in size in the beginning of compression and in the later part decrease significantly. It was also found that compression predominantly increased axial fluctuations during compression.
CHAPTER 8

Future work

*Le cadavre exquis boira le vin nouveau*
Duhamel, M., Prévert, J. and Tangy, Y.

The aim of all work presented in this thesis is to increase the knowledge of in-cylinder flows. However, several steps remain in order to properly analyze and understand flows created during intake and the effect of compression. The comparison between steady flow rig and simulations are somewhat unsatisfactory with relatively large differences. Primary results suggest that a major contribution to the discrepancies is the difference in outlet geometry between the two cases. There is also a lack of knowledge how a flow field such as the two counter rotation vortices found for 5 mm lift is affected by compression. Furthermore, the swirl number used to characterize the engine is a weighted mean of the (mean) swirl numbers measured in the steady swirl test rig. The rationale behind this is a quasi-steady assumption that the speed of the valve is sufficiently low not to affect the created flow field.

During compression the quantitative representations of a flow field consists of swirl and tumble number and in some studies fluctuations from the cycle mean. Hence, neither description of distribution of turbulence nor any length scales. Swirl and tumble number assumes a solid body type rotation which is not necessary the case as different port geometries with equal swirl numbers has been shown to produce vastly different emissions, Miles (2009).

The planned future work can be summarized as follows:

- Validity of steady flow rig results, i.e. what flow quantities found in a steady flow rig can be found in an engine. This includes a more detailed sensitivity analysis of outflow geometry.
- Study the validity of the quasi-steady assumption used when measuring the swirl and tumble numbers.
- Effect of accelerating flow on test rig swirl numbers.
- The effect of piston bowl on in-cylinder flow quantities.
- Effect of pressure pulsations on the inlet conditions on in-cylinder flow structures.
8. FUTURE WORK

- Investigating the hypothesis made by Bensler et al. (2002) regarding the effect of unstable test rig swirl number on cycle-to-cycle fluctuations.
CHAPTER 9

Papers and authors contributions

Paper 1
*A Coupled PIV-LES Approach to Understand Port Generated Structures*
Martin Söder, Julie Vernet, Björn Lindgren & Laszlo Fuchs, 2013, Presented at LES4ICE

The purpose of this paper was to compare and explain the results from PIV measurements and LES simulations in a steady swirl test rig. 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
*Study of in-cylinder flow structures using a coupled PIV-LES method*
Martin Söder, Julie Vernet, Lisa Prahl Wittberg & Laszlo Fuchs, 2013, in process to be submitted

Further description and processing by the results of PIV measurements and Large-eddy simulations of the flow in a steady swirl test rig. LES simulations were performed by Söder under supervision by Prahl Wittberg and Fuchs. PIV measurements were performed by Vernet under supervision by Söder and Lindgren. To be submitted.

Paper 3
*Compression of a swirling and tumbling flow*
Martin Söder, Lisa Prahl Wittberg & Laszlo Fuchs, 2013, Accepted to 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.

**Paper 4**

*Effects of compression on coherent structures in an enclosure*

Martin Söder, Lisa Prahl Wittberg & Laszlo Fuchs, 2013, Submitted to ICJWSF2013

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.
Acknowledgements

First of all I would like to thank my supervisor Laszlo Fuchs for all encouragement, guidance and support. My second supervisor Lisa Prahl Wittberg is acknowledged for her many comments on my articles. I would also like to thank the members of the project steering committee Björn Lindgren, Niklas Nordin, Dennis Konstanzer, Raymond Reinman, 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 and the department of Mechanics for all nice Fikas and discussions. Without you my day would have been a lot less fun.

Last, but definitely not least, I would like to thank Annika and my family for love and support.
References

2010 World Energy Outlook. IEA.


Anderson, J. 1995 Computational fluid dynamics.


Buschbeck, M., Bittner, N., Halfmann, T. & Arndt, S. 2012 Dependence of combustion dynamics in a gasoline engine upon the in-cylinder flow field, determined by high-speed PIV. Experiments in Fluids pp. 1–12.


Dean, W. & Hurst, J. 1959 Note on the motion of fluid in a curved pipe. Mathematica 6, 77–85.

REFERENCES


DIESELNET 2013 Emission Standards, European Union.


REFERENCES


PATNAIK, G., BORIS, J. P. & GRINSTEIN, F. F. 2003 Large scale urban simulations with the MILES approach. In AIAA CFD Conference, Orlando Fl.


PRASAD, A. K. 2000 Stereoscopic particle image velocimetry 29 (February).

REFERENCES


THIEN, G. 1964 Description of measuring methods for the investigation of stationary flow properties at valve ports and statements for the evaluation of the measurement results. Tech. Rep., AVL List GmbH.


VERMOREL, O., RICHARD, S., COLIN, O., ANGELBERGER, C., BENKENIDA, A. & VEYNAME, D. 2009 Towards the understanding of cyclic variability in a spark ignited engine using multi-cycle LES. Combustion and Flame 156 (8), 1525–1541.


WORLD BANK 2013 Data the World Bank.