Reduced Models for Flows in IC-Engines

by

Niklas Winkler

October 2011
Technical Reports from
Royal Institute of Technology
KTH Internal Combustion Engines
SE - 100 44 Stockholm, Sweden

©Niklas Winkler 2011
Abstract

The finite response time of the turbocharger is the most notable effect of transient operation on a turbocharged diesel engine. To fulfill future emission requirements, high amounts of transient EGR will be required even though after-treatment devices are being used. This implies that advanced turbocharger systems have to be introduced to enable high boost pressure with improved or at least maintained response time. The increased amount of tuneable parameters from the more advanced turbocharging system will make it difficult and expensive to optimise the engine experimentally. Therefore, the wish is to optimise the engine design using numerical tools. This requires predictive models for the gas exchange system and its components, i.e. the turbocharger, the manifolds and the cylinder with its valves. The results have shown that engine simulation tools based on tabulated data from measurements can qualitatively predict the engine’s performance and be used as a first step in engine design. However, these models have to be calibrated extensively in order to reflect the engine performance quantitatively. The objective of this research is therefore to find a modelling approach for the gas exchange system and its components that are more predictive than the tools used, but still computationally effective enough to be suited for engine simulations.

The thesis contains a summary of on-engine experimental results as well as some One-Dimensional (1D) simulations. Since the 1D modelling approach has limited range of validity and applicability, we have considered two approaches for reducing the full Three-Dimensional (3D) governing equations, which are a set of Partial Differential Equations (PDE), in a systematic manner. The first approach is based on a numerical length scale analysis of the different terms in the governing equations after changing the coordinate system with one coordinate aligned with the flow path. By retaining the most important terms or neglecting the (significantly) smallest terms, different reductions may be attained, which in their simplest form may look like the boundary layer equations. The results for a double bent pipe, used to illustrate the approach, show that the most significant component of the viscous terms is the radial component, which is in the order of two magnitudes larger than the axial and azimuthal components. The convective terms are all in the same order of magnitude, whereas the radial component is of significant importance in the bends of the pipe due to centrifugal forces, and the azimuthal component after the second bend due to a swirling motion. For the flow in a straight pipe, the approach would give the same model as the common 1D simulation tool. However, for pipes with a more general shape, the approach is superior as it allows for a rational reduction of the governing equations. The main limitation of the approach is for flow situations that do not have a dominating flow direction. Under such conditions the a priori and/or the a posteriori analysis would reveal that the reduction is inconsistent. Thus, the approach
implies maintained efficiency, but with improved (and assessable) accuracy as compared to the common 1D simulation tool.

The second approach is based on the Galerkin projection of the governing equations projected onto Proper Orthogonal Decomposition (POD) modes. These POD modes are computed for the flow in a given geometry obtained through Large Eddy Simulations (LES). The Galerkin projection results in a system of Ordinary Differential Equations (ODE) for the time-dependent coefficients of the 3D POD modes. The results show that the method is best suited for flows with strong coherent structures. However, the system of ODEs may be inherently unstable (depending on the number of modes used in the simulation) and modelling errors grow with each time-step. This limitation may be remedied by numerically preventing any exponential growth. The approach can also be extended into a combined Galerkin (ODE) - LES (PDE) approach by replacing several steps of the LES by the same amount of time steps with the reduced model. It should be pointed out that this approach can provide the full 3D flow field in contrast to space reduced models (e.g. the common 1D tool) and thereby handle more complex flows with high degree of computational efficiency.

To summarize, the thesis demonstrates new possibilities of obtaining reduced models suited for engine simulations based on 3D CFD. With this application in mind, these tools are novel and their evaluation and assessment should be extended to other components of the gas exchange system.
"The purpose of computing is insight, not numbers"
Hamming’s motto
CHAPTER 1

Introduction

For heavy-duty applications as road-going transportation vehicles the diesel engine is today the only realistic option due to its high efficiency, reliability and cost. Even though engine development the past years has been focusing on reduction of emissions, the road transportations are the largest contributor to nitrogen oxides ($NO_x$) and the second largest contributor to particle matter ($PM$), see e.g. Laguna-Gomez (2007). The primary emission reduction mechanisms for $NO_x$ is Exhaust Gas Recirculation ($EGR$) and for $PM$, developments of the injection system including considerably elevated injection pressures, even though aftertreatment devices are today widely used. As higher amounts of $EGR$ are introduced, higher boost levels are needed to maintain the oxygen content.

The purpose of turbocharging has in the past been to increase the power to weight ratio of the engine. By increasing the amount of air available for the combustion process, more fuel can be burned effectively. Today, the primary goal to use turbocharging for heavy-duty applications is to be able to optimize the engine with respect to emissions in order to manage future emission legislations while maintaining or even improving fuel efficiency.

The turbocharger itself has become a mature product. Improvements on the turbomachine will therefore be rather incremental and the focus for development has turned more towards the turbocharger application and its integration into the Internal Combustion (IC) engine gas exchange system. New complex systems to improve exhaust energy utilisation over a wide engine operating range will be more frequently incorporated into the engine design such as Variable Geometry Turbines (VGT), multi-stage turbocharging, turbo compounding or various additional mechanically driven compressors etc.

The main challenge for engine manufacturers is to be able to follow the engine transients, with a time scale in the order of a second, in the transient emission certification cycles as the European Transient Cycle (ETC) without exceeding the emission limits. For turbocharged diesel engine operation the most notable effect of transient response is turbocharger lag due to the relatively long response time of the turbocharger. This is caused by the shaft moment of inertia of the turbocharger. The change in energy delivered by the engine to the turbine must first accelerate the turbocharger shaft assembly
INTRODUCTION

To make it possible for the compressor to produce boost pressure, the turbocharger system is in focus for overall engine performance. With the more complex turbocharging systems the amount of tuneable parameters increases, which makes it difficult to optimize the engine experimentally. Therefore, the wish is to be able to optimize the engine through computer simulations.

The flow in the engine gas exchange system, including the turbocharger, the manifolds and the cylinder with its valves, can be described by basic conservation laws of mass, momentum and energy. These conservation laws may be formulated as a system of Partial Differential Equations (PDE), which is nonlinear and requires appropriate boundary conditions for admitting a solution. Since the flow is turbulent and contains a large range of scales, which one cannot resolve, one has to model the turbulence. In-spite of using simplifying models, the solution of the time-dependent flow still requires a considerable computational power, which for the time being prevents the use of such methods, i.e. Computational Fluid Dynamics (CFD) for a complete engine, even for research purposes. CFD methods are currently used for analysis of the flow in the different components of the engine and modelling of the system is handled by assuming a One-Dimensional (1D) flow throughout the engine. Thus, engine simulation tools are based on the 1D governing equations, which assume that the flow is uniform across the flow direction. To account for the deviation of the 1D model from the real flow, one introduces so called losses, a source term that is added to the momentum equation. This term is based on a pressure loss coefficient obtained from measurements. The pressure loss coefficient is used for each part or segment corresponding to the different engine components, and thus the model cannot consider secondary flow effects.

To model the turbocharger components as the turbine or compressor, data from stationary flow rig measurements are used as a look up table with the assumption that the turbine works under quasi-steady flow conditions. Another issue is related to the fact that it is difficult to accurately measure the pressure drop and efficiency at low mass flow rates. Thus, the set of experimental data is limited to a small part of the operational envelope of the turbocharger in real engine applications. This fact has stimulated the development of different models for predicting the performance of turbochargers. Unfortunately, the existing models have clear limitations, which also motivate the work done within this thesis.

1.1. Objectives

This thesis is divided into two major parts. The objectives of the first part are to gain knowledge of how complex turbocharger systems perform on the engine and to study the possibilities to simulate engines with complex turbocharger systems in transient operation using a commercial engine simulation software. The objective of the second part of this research is to find new approaches to
derive reduced models for the flow in the gas exchange system of IC-engines that are more accurate than the models used.

1.2. Methods
In the first part of the thesis a heavy-duty diesel engine with two different turbocharger system configurations was simulated in a commercial quasi-1D engine simulation code. The engine was also run in a test-cell at KTH in order to gather data for validation of the simulation results. In the first configuration the engine was equipped with a single turbocharger system including a twin-entry turbine. In the second configuration the engine was rebuilt with a two-stage turbocharger system in series, which comprises a twin-entry turbine at the high pressure stage, a VGT at the low pressure stage and short route EGR. As a complement to the turbine modelling via measured turbine performance maps, a commercial turbine modelling tool based on semi-empirical relations was used to produce turbine performance data.

In the second part of the research two different approaches have been considered in order to derive reduced models for engine simulations. The first method is aimed at reducing the governing equations in a systematic manner, which enables an a priori and an a posteriori error estimation of the reduction. The governing equations are projected onto the flow path and the different terms are quantified. The analysis is based on Three Dimensional (3D) CFD data for a specific geometry. The governing equations can then be reduced by excluding the non-dominating terms. The advantage of this method is that an error estimation can be obtained, which indicates the accuracy of the approximation when using the reduced model. The disadvantage is that a 3D computation on the specific geometry has to be performed as a basis for the method instead of the conventional approach where tabulated (experimental) data for each part is used. The second approach is based on utilizing the Proper Orthogonal Decomposition (POD) of a full 3D flow field. POD is a decomposition method, which decomposes the flow field into a spatial and a temporal part. By projecting the governing equations onto the POD modes, the equations can be reduced to a set of Ordinary Differential Equations (ODE) in time. As an input to the model one needs a set of Large Eddy Simulation (LES) or detailed experimental data, which constitutes the basis of the POD modes. Hence, the modelling approach allows continuing the time marching of detailed LES computations in the parameter space of the problem.

1.3. Thesis Contributions
The main contributions from the first part of the thesis are as follows,

- The range of the turbine performance maps can be extended with semi-empirical models from the open literature to allow engine simulations at their full operating range.
1. INTRODUCTION

- A novel design of a two-stage turbocharger system was developed for model validation, which showed promising results with respect to engine performance.

The contributions of the second part of the thesis are:

- The governing equations can be reduced systematically by a length scale analysis in a problem specific manner. The method enables an a priori and an a posteriori error estimation of the reduced model.
- Proper orthogonal decomposition of the 3D flow allows improved understanding of the flow structures of the pulsatile flow. Additionally, the POD basis has been used to reduce the governing equations via Galerkin projection onto POD modes to obtain a reduced ODE based model for engine simulations.
CHAPTER 2

Turbocharging

Turbocharging dates back to the time when the IC-engine was first developed. Gottlieb Daimler and Rudolf Diesel thought of increasing their engine's performance by pre-compressing the available air for combustion.

The first successful application of a turbocharger to compress air was around the year 1918 when Dr. Sanford A. Moss began developing turbochargers to allow airplanes to fly higher than before. The turbocharger built was attached to a V12 Liberty aircraft engine and tested at Pikes Peak, i.e. at 4300 metres of height. Dr. Moss showed that his engine could eliminate the power losses experienced at high altitudes due to the reduced density. In 1925, Dr. Alfred Büchi successfully applied a turbocharger to an engine and managed to achieve an engine power increase of 40%. The first turbocharged engine for trucks was built by the Swiss Machine Works Saurer in 1938.

Traditionally turbocharging has been associated with high power improvements but suffered from problems as reliability and turbocharger lag. Continuous development and new techniques have solved some of the past problems. Today, especially for road transport applications, turbocharging is more and more becoming a help to optimise engines for improved fuel efficiency and to reduce harmful emissions. However, optimizing a turbocharger to a specific engine is still a challenging task due to their different flow characteristics.

2.1. General Principles

The typical turbocharger in an automotive application is a simple device, which is mechanically separated from the engine. The turbocharger consists of a centrifugal compressor, which is mechanically connected to and driven by a single radial turbine, Figure 2.1. The turbine extracts energy from the otherwise wasted exhaust gases. The role of the compressor is to increase the air density entering the combustion chamber, thus a higher available mass of air is achieved for the combustion process. In small engines as those used for passenger cars or trucks, radial turbo machines are mostly used since their efficiency is superior to corresponding axial turbo machines due to their large tip clearance losses and bad blade to height ratio.

Even though the turbocharger itself is a simple device, the design process is complex due to the unsteady environment when employed on a reciprocating
internal combustion engine. The engine exhausts are characterised by short and long scale transients, the former driven by exhaust valve events and the latter by engine operating conditions. The complexity comes from that the long scale transients makes the turbocharger work in a wide range and the short scale transients makes the on-engine turbocharger performance analysis complex.

How well a turbocharger works on a given engine can be measured as turbocharger efficiency ($\eta_{TC}$), which is the product of compressor efficiency ($\eta_C$) and turbine efficiency ($\eta_T$) including losses in the drive system ($\eta_{mech}$) of the connecting shaft according to,

$$\eta_{TC} = \eta_C \eta_T \eta_{mech}$$  \hspace{1cm} (2.1)

The isentropic efficiency of the turbine and compressor is defined as,

$$\eta_T = \frac{W_{extr}}{W_s}$$  \hspace{1cm} (2.2)
2.3. Matching Turbocharger and Engine

\[ \eta_C = \frac{W_s}{W_{\text{extr}}} \]  

where \( W_{\text{extr}} \) denotes extracted work and \( W_s \) work at isentropic conditions. The presence of a turbine in the engine exhaust system increases the engine back pressure, thus the engine pumping losses increases. The compressor on the other hand raises the intake air pressure, which decreases the pumping losses. Therefore, to gain in engine efficiency due to the turbocharger the overall turbocharger efficiency has to be high enough. This is difficult to obtain for an entire engine operational range with a conventional turbocharger. The problems with turbocharger to engine matching are discussed in Section 2.3.

2.2. Turbocharging Systems

Ideally there are two main methods to extract the exhaust energy, the constant pressure and the pulse pressure system. In a constant pressure system the exhaust manifold includes a large enough volume to dampen out the pulsations from the engine. The theoretically available energy for this type of system is shown by the p-V diagram of the idealized engine cycle bounded by the points 7-8-10-11 in Figure 2.2. In the pulse pressure system an attempt is made to extract energy associated with the high pressure pulsations from the opening of the exhaust valve including the energy available for constant pressure turbocharging. To preserve the kinetic energy of the exhaust pulses an exhaust manifold which is as short and narrow as possible is used. Ideally, the available energy for this type of system is shown by the p-V diagram bounded by the points 5-8-10-11-13 in Figure 2.2.

The pulse pressure system makes more exhaust gas energy available for the turbine than the constant pressure system, but with a pulse pressure system it is more demanding to achieve a high mean turbocharger efficiency over the engine cycle. The unsteady flow makes the turbine work in off-design operation for a large part of the engine cycle, see e.g. Watson & Janota (1982), which makes it more challenging to analyse the turbine performance and hence to optimize its design. For automotive applications pulse pressure turbocharging is used since it has shown higher potential with respect to exhaust gas energy utilization. To prevent exhaust pulses to interfere for multi-cylinder engines twin-entry volutes are used. A twin-entry volute, which has its channels divided axially is situated on the radial turbine shown in Figure 2.1.

2.3. Matching Turbocharger and Engine

Due to the different flow characteristics of the IC-engine and the rotor dynamic machines, i.e. the turbine and compressor, matching a turbocharger to an engine is a difficult task. The internal combustion engine works like a positive displacement pump where its volumetric flow rate increases close to linearly
with engine speed. The operational range for a heavy-duty diesel engine is from 600 to 2000 rpm while smaller engines can work in an even broader operational range, from 1000 rpm to 6000 rpm. The desire is to obtain high inlet air density over the entire range so an engine of smaller size can be used without sacrificing power output and will lead to a gain through less frictional losses.

The turbine has similar characteristics as a nozzle, i.e. the pressure ratio across the turbine is proportional to the density and the square of the flow velocity, $\Delta p \propto \rho u^2$. Thus the limitations for a turbine are that at low engine speeds, i.e. low mass flow rates, the expansion ratio across the turbine is small and hence the power output. At high flow rates the turbine reaches a flow limit where it becomes choked, i.e. the velocity within any part of the volute or rotor becomes sonic, thus the pressure upstream the turbine rises with only a minor increase in mass flow rate. Figure 2.3a shows the flow characteristics of a radial turbine.

The volumetric flow rate through the compressor is controlled by the inlet area and the velocity of the flow into the impeller. The rotational speed of the impeller controls the tangential velocity of the flow, thus the power input which is limited by the maximum speed of the compressor due to limitations of impeller material properties. Since the rotational speed does not directly influence the flow velocity onto the impeller, its operating flow range is relatively narrow, Figure 2.3b. As shown by Baines (2005) the flow range for a centrifugal compressor is exponentially decreasing with its working range with respect to pressure ratio. The flow range is limited by surge and choke. Surge can occur at low flow velocities into the impeller due to flow separation at the

**Figure 2.2.** The Seiliger cycle representing the ideal IC-engine cycle.
impeller blades and at high flow rates the velocities within the impeller may become sonic and choke will occur.

A general limitation of the turbocharger in an automotive application is its transient response, i.e. to follow the load changes of the engine. The time it takes to reach a certain turbocharger speed and produce boost is caused by the shaft moment of inertia of the turbocharger. The energy delivered to the turbine must first accelerate the turbocharger shaft assembly to make it possible for the compressor to produce boost pressure. The time it takes for the turbocharger to spin up is called turbocharger lag. Figure 2.4 shows a typical example of the spin up process of a turbocharger for an instant load step demand. The compressor will produce the desired boost pressure after approximately two seconds in this case, i.e. after the power output of the turbine and the compressor power and the mechanical losses are in equilibrium, which is when the turbocharger speed to reaches a steady state.

To improve its transient response and low speed performance a smaller turbocharger can be used, but it will become choked for high engine speeds. On the other hand a large turbocharger will give good performance at high engine speeds, but due to its high inertia and the higher flow rate needed for the same boost pressure it will give poor transient response and low speed performance. At too low engine speeds the exhaust flow will not be sufficient for the turbocharger to produce boost pressure, i.e. there is a boost threshold speed, which should not be confused with turbocharger lag. A way to improve these problems is to use variable geometry turbines and/or multi-stage
2. TURBOCHARGING

Figure 2.4. Turbocharger speed on an engine during a load transient for a heavy-duty engine, i.e. a load step demand at constant engine speed.

turbocharging, which have in many ways improved the turbocharger to engine matching possibilities.

2.3.1. Variable Geometry Turbine

The VGT is considered as an effective approach to the turbocharger to engine matching problem. By varying the inlet turbine geometry it is similar as having a finite range of turbine sizes in one unit. At low engine speeds the VGT is kept closed to raise the pressure upstream the turbine, thus the exhaust gas energy is raised. At high engine speeds the inlet area is increased to avoid over boosting and high engine back pressure. The inlet area is mainly adjusted in two ways, by a sliding nozzle ring mechanism, Figure 2.5 or by pivoting nozzle blades, Figure 2.6. The sliding nozzle ring mechanism keeps the nozzle blades fixed and changes the inlet area due to an axial movement of the sliding wall. By pivoting the nozzle blades the area between the blades changes as well as the vane angle, which affects the incidence angle, i.e. the angle of the flow onto the rotor blades.

2.3.2. Two-Stage Turbocharging

Two-stage turbocharging is an effective way of overcoming the limitations on boost pressure imposed by current compressor materials, to extend the operational range and to improve the transient response. Already for a boost pressure of 1.5 bar and an assumed compressor efficiency of 70 % and with an inlet temperature of 25°C, the outlet temperature of the compressed air reaches 150° C. At these temperatures the compressor impeller materials needs to be more exotic than the normally used aluminium alloys, since its material properties start to decrease significantly. For high boost pressures, the compressor
2.3. MATCHING TURBOCHARGER AND ENGINE

Figure 2.5. VGT with a sliding nozzle ring mechanism, closed and open position respectively, from Cummins Turbo Technologies.

Figure 2.6. VGT with variable nozzle blades, from Heizler (1995)

impeller is manufactured from different titanium alloys that can withstand the higher temperatures.

In a two-stage turbocharging system an intercooler can be used between the compressor stages, which reduces the inlet air temperature to the high pressure stage, thus making it possible to reach high boost pressures with less exotic impeller materials. An increased air density, will also give a lower volume
flow rate through the compressor and hence less power is needed to reach the desired pressure raise.

In a sequential turbocharger arrangement two or several different sized turbochargers can be used where the flow can be switched so that the best suited turbocharger for that operational point is used. Sequential turbocharging can improve low end torque, response time and/or operational range. Sequential turbocharging can nowadays be found on several premium passenger car models.

However, there are down sides with two-stage turbocharging, such as added complexity to optimise the gas exchange system, packaging issues in a vehicle and the extra cost.

2.4. The Turbocharger

Ideally the turbo machine works with the principal described by the Euler turbo machinery equation, Moustapha et al. (2003), which describes the work transfer within the rotor with the assumption that the heat transfer is negligible. The Euler equation is derived from Newton’s second law of motion applied to a rotating system,

\[ W_x = \Delta E_{k,x} = \tau \omega / \dot{m} = \omega \left( r_1 C_\theta_1 - r_2 C_\theta_2 \right) = (h_{0.1} - h_{0.2}) \]  \hspace{1cm} (2.4)

where \( W_x \) denotes the work transfer, i.e. the rate of kinetic energy change per unit mass flow rate denoted \( \Delta E_{k,x} \). \( \tau \) denotes the generated torque, which is the change in angular momentum, i.e. the change in radius times the change in tangential velocity \( C_\theta \) onto the rotor. The work per unit mass flow rate, which is equal to the change in total enthalpy, \( h_0 \) of an adiabatic process is thus proportional to the temperature change across the turbo machine.

2.4.1. The Radial Turbine

The radial turbine consists of a volute casing, a set of vanes, which are usually omitted, the turbine housing and the turbine rotor. The purpose of the volute is to expand the tangentially incoming flow to a high flow velocity and to distribute the flow circumferentially around the rotor to achieve a uniform mass flow and static pressure at the volute exit. The vanes (if present) then further accelerate the flow, reducing pressure, increasing kinetic energy and turn the flow to the designed angle. In the rotor the incoming flow is expanded through the rotor passages and the kinetic energy is extracted and further transferred to mechanical energy.

Expanding the Euler equation according to the gas velocity components at the rotor inlet and the rotor exit, notations according to Figure 2.7, gives design criteria for the rotor and volute,
2.4 THE TURBOCHARGER

Figure 2.7. Velocity relations at the rotor inlet, left and exit, right. The angle relative to the rotor, $\beta$ is defined to be negative.

$$W_x = \frac{1}{2} \left[ (U_2^2 - U_3^2) - (W_2^2 - W_3^2) + (C_2^2 - C_3^2) \right]$$

(2.5)

where $W$ denotes the flow velocity relative to the rotor, $C$ the absolute flow velocity and $U$ the blade tip speed. In a radial turbine the change in inlet and exit radius gives a change in blade speed such as $U_2^2 > U_3^2$. The second term which relates inlet and exit relative flow velocities shows that acceleration of the flow within the rotor passages is desired. To maximise the third term a high inlet absolute flow velocity from the volute and a minimum rotor exit absolute flow velocity is desired since the kinetic energy of the flow will be dissipated. According to Japikse & Baines (1997) there does not exist any analytical way to select an exit radius of the rotor to optimise the energy transfer, only conventions.

The ratio of expansion between the volute and the rotor is denoted as reaction ($\Lambda$) and defined as the enthalpy change in the rotor and the total enthalpy change across the stage, i.e. from volute inlet to the rotor exit,

$$\Lambda = \frac{\Delta h_{rotor}}{\Delta h_{0,stage}}$$

(2.6)

For an isentropic radial inflow turbine with complete exhaust energy recovery and a reaction of 50%, which is common design practice, maximum efficiency occurs ideally at a blade speed ratio of $1/\sqrt{2}$ according to Watson & Janota.
(1982), i.e. the ratio between the rotor inlet blade tip speed ($U_2$) and the gas velocity at isentropic expansion ($C_s$) across the stage,

$$\frac{U_2}{C_s} = \frac{r (2\pi N/60)}{\sqrt{2c_pT_{00} \left[ 1 - \left( \frac{p_3}{p_{00}} \right)^{\frac{\gamma-1}{\gamma}} \right]}}$$

where index 00 denotes the total turbine inlet state and $N$ the rotational speed.

Figure 2.8 shows an example of measured efficiencies at different constant rotational speeds as a function of blade speed ratio. The left part of the efficiency parabola relates to high gas velocities and the right to lower gas velocities in relation to the wheel speed.

The angle at which the flow approaches the rotor relative to the blade angle is called the incidence angle,

$$i = \beta_2 - \beta_{2b}$$

where $\beta_2$ denotes the angle at which the flow approaches the rotor, defined as negative in Figure 2.7, and $\beta_{2b}$ denotes the inlet blade angle, which is usually zero. The incidence angle has a significant effect on the efficiency of the rotor. As claimed by Spence & Artt (1998) this is the primary cause of reduced efficiency at off-design operation. At a non optimal inlet incidence angle work has to be put in to turn the flow according to the blade passages. A flow visualization is shown in Figure 2.9 where the flow is approaching a rotor at different incidence angles. The visualisation shows that a uniform flow without
recirculation occurs for a somewhat negative incidence angle. This is due to that the radius diminishes more rapidly than the tangential velocity, which causes a circulation of the flow to the pressure side where it stagnates. Thus the flow will first approach the suction side and then the pressure side. At incidence angles away from the optimal angle the flow will separate at the suction respectively at the pressure side.

The incidence angle changes with rotational speed and/or pressure ratio across the stage. An increase in rotational speed will give a negative change in incidence angle and the opposite for a decrease in speed. An increase in pressure ratio will give a positive change to the incidence angle and vice versa for a decrease in pressure ratio. Moustapha et al. (2003) shows how changes in incidence angle change the turbine performance, Figure 2.10. From the design point at maximum efficiency the performance is more affected to a decrease than to an increase in rotational speed, thus to positive changes in incidence angle. The turbine performance will also be affected by changes in pressure ratios, to a lower efficiency as the pressure ratio increases for relatively large pressure ratios and vice versa for relatively small pressure ratios across the turbine. Moustapha et al. (2003) also states that an optimal incidence angle can be found in the region of $-20$ to $-40^\circ$. Yeo & Baines (1990) measured the incidence angle for a turbine at its maximum efficiency point, which showed that the optimum incidence angle is close to $-30^\circ$.

Apart from incidence losses there are passage losses within the turbine. Passage losses include losses due to secondary flows, blockage and loss of kinetic energy due to growth of boundary layers in the volute and the rotor passages to the rotor throat. From the rotor throat to the space downstream the trailing
Figure 2.10. Influence of incidence angle on turbine performance. The dots denote measured data points.

edge there are losses due to the velocity field distribution downstream of the rotor. The trailing edge losses are influenced by the rotor exit deviation angle,

\[ \delta = \beta_{3b} - \beta_3 \]  

(2.9)

where \( \beta_{3b} \) denote the exit blade angle and \( \beta_3 \), the exit flow angle. As previously pointed out from the expanded Euler equation, Equation 2.5, a low absolute flow velocity is desired at the rotor exit and a large flow velocity relative to the blade angle, which is a contradiction. The velocity triangle for the rotor exit shows that the relative flow angle, thus the deviation angle, is related to the blade tip speed and the tangential (\( C_{\theta} \)) and the meridional (\( C_m \)) component of the absolute flow velocity. Unfortunately, there is not much data published on the deviation angle at the optimum design point. Moustapha et al. (2003) summarised a few measurements where it can be seen that the deviation angle deviates from -7 to +2 degrees at the optimum design point. Baines & Yeo (1991) measured the absolute flow angle at the design conditions and at lower blade speed ratios. The measurements showed that the absolute flow angle was nearly constant across the rotor exit at the design point but deviates significantly at lower blade speed ratios due to that the flow velocity becomes higher near the rotor hub. For the twin-entry turbine used the exit absolute flow angle was not affected by different partial admission conditions.
2.4. THE TURBOCHARGER

2.4.2. Twin-Entry Turbine Performance

On multi-cylinder engines, twin-entry turbines are used to prevent exhaust pulses from different cylinders to interfere. This prevents exhaust gases to flow backwards into the cylinders and eventually to the intake side of the engine at valve overlap. Another aspect is that the exhaust pulse energy is retained, which e.g. improves the transient response of an engine.

Several researchers have been measuring the twin-entry turbine performance with different degrees of unequal admission conditions in flow rigs. Dale & Watson (1986) showed that the mass flow characteristic for each branch of a symmetrical twin-entry turbine at equal pressure ratios is half of the combined flow. However, at unequal admission they concluded that the mass flow characteristics for each branch cannot be accurately represented by halving the total flow at equal admission and applying the results to each branch in isolation of the other since the flow in each branch is affected by the pressure ratio in the other branch. They also showed that the efficiency differs with different unequal admission conditions, +5 to -12% from the steady equal admission line as shown in Figure 2.11. Capobianco & Gambarotta (1993) performed similar measurements that supported the findings even though their measurements were performed on a twin-entry turbine with asymmetrical volute limbs.

Yeo & Baines (1990) extended the measurements performed by Dale & Watson (1986) with measurements of the full velocity vector from which the flow angle and incidence angle were derived. The measurements showed large variations in incidence angle even at close to design point operation for unequal admission conditions, which affects the turbine performance. Unequal admission conditions were further investigated by Baines & Yeo (1991) who showed that at equal admission the incidence angle is close to −30° at different turbine operating points, but differ substantially for different unequal admission conditions, ±15° for modest degrees of unequal admission (mass flow rate ratio
of 0.6 to 2.0) and up to $-60^\circ$ to $+30^\circ$ for extreme unequal admission conditions. At these extreme conditions the flow angle was measured to be greater than $90^\circ$ near the walls, which implies that recirculation to the other volute limb occurs. The measurements support the 3D simulation results by Lymberopoulos et al. (1988) of the flow within a twin-entry volute.

2.5. Turbine Performance under Pulsating Flow Conditions

A significant amount of work has been performed in the field of radial turbine performance under pulsating flow conditions to assess the behaviour of the turbine in engine like conditions. The studies have primarily been performed in pulsating flow rigs to get controlled boundary conditions and the possibility to use measurement apparatus with high accuracy, which would not be applicable for real engine conditions as e.g. delicate hot-wire probes. On-engine experiments have however also been conducted with a full set of measured parameters.

The flow in an IC-engine is nominally unsteady and covers a large range of frequencies. The low frequency unsteadiness is on the engine level and often termed as engine transients from changes in power demand, rack setting in a VGT or opening and closing of the EGR valves. The frequencies corresponding to the engine events are on the order of 0.1-1 Hz. The second level of unsteadiness is on the engine to turbocharger level due to exhaust pulsations from the opening and closing of the valves. The corresponding frequencies are on the order of 10-100 Hz, which are the most relevant frequencies for assessing the on-engine turbine performance. The last level of unsteadiness is due to rotor blade passages at the volute tongue, which are on the order of 10000 Hz. A criterion for assessing the importance of the unsteadiness in a flow situation is the Strouhal number, $St = fD/U$, which relates the flow unsteadiness to the flow velocity. Hence, for a low Strouhal number, $St << 1$ the flow situation can be seen as quasi-steady, i.e. stationary at every instance in time. Costall (2007) computed the Strouhal number for the engine to turbine unsteadiness for a four-cylinder passenger car diesel engine with an engine speed of 1000-5000 rpm to 0.10-0.33. Hu & Lawless (2001) computed the Strouhal number for a six-cylinder heavy-duty diesel engine to 0.4 at an engine speed of 2000 rpm. Hence, dynamic effects could be of significant importance for the on-engine turbine performance.

The first investigation found of pulsating flow performance of radial turbines was conducted by Wallace & Blair (1965). Even though they could only measure the static pressure instantaneously, they found strong indications that the unsteady behaviour of the turbine is more apparent as the pulse frequency increases. In the same year Benson & Scrimshaw (1965) performed experiments to compare turbine performance under steady and pulsating flow conditions for a nozzled radial turbine. The measurements showed that the output power and swallowing capacity of the turbine under pulsating conditions was higher than
under steady flow conditions for nearly all the operational points measured. The magnitudes of these effects were dependent on the pulse frequency and turbine speed. Furthermore, the efficiency of the turbine was higher under pulsating than under steady flow conditions. Both investigations indicated that a quasi-steady approach will most likely not be able to predict turbine performance under pulsating flow conditions.

Dale & Watson (1986) presented the first turbine performance measurements found under pulsating flow conditions with a complete set of instantaneous measured parameters. The measurements were conducted on a cold flow rig with a mean total inlet air temperature of 400 K. The parameters measured instantaneously were mass flow rate air measured with hot-wire anemometers, static pressures with strain gauge pressure transducers and instantaneous turbine torque, \( \tau \), as a sum of the mean and fluctuating torque computed from the moment of inertia of the shaft, \( I \) and the change in angular velocity \( \omega \) of the shaft,

\[
\tau = \tau + I \frac{d\omega}{dt} \tag{2.10}
\]

The time-averaged torque \( \tau \) was measured with a dynamometer and the rotational speed was measured via a crystal clock with a frequency of 50 MHz trigged by six positional datums every shaft revolution. These measurement techniques have later been used by several researchers and are still used as e.g. by Rajoo (2007). Dale & Watson (1986) and later Dale (1990) showed that the instantaneous efficiency, Equation 2.11, as a function of mass flow rate under pulsating flow conditions deviates from the efficiency measured under steady flow conditions at the same pressure ratio, Figure 2.12. The efficiency was derived for the twin-entry turbine used as the ratio between the shaft power and the sum of the isentropic power output for each volute inlet,

\[
\eta_{TS} = \frac{\dot{W}_{\text{extr}}}{\dot{W}_{si} + \dot{W}_{so}} \tag{2.11}
\]

where the extracted power is the power used to compress the air and the power to accelerate and decelerate the turbo shaft assembly,

\[
\dot{W}_{\text{extr}} = \left( \frac{2\pi}{60} \right)^2 J_{\text{rotor}} N_{\text{turbo}} \frac{dN_{\text{turbo}}}{dt} + \dot{m}_{\text{air}} \int_{T_{i1}}^{T_{o2}} c_{p,\text{air}} dT \eta_{\text{mech}} \tag{2.12}
\]

and the power from isentropic expansion across the turbine for each volute entry, i.e. the outer and inner entry denoted \( o \) and \( i \), where the inner entry is the entry closest to the compressor,
The turbine efficiency for a pulsating flow condition was shown to be below the steady flow line, which contradicts the findings by Benson & Scrimshaw (1965) while the instantaneous mass flow rate was shown to be close to the steady flow line.

A similar investigation was performed by Capobianco et al. (1989) who used a small size automotive turbocharger to measure the power output from the turbine under pulsating flow conditions. The result was that the swallowing capacity was 3-6% higher than measured under steady flow, but the efficiency was significantly higher, 10-20%. However, the power measurements were derived from compressor performance measurements without the possibility to measure the instantaneous turbocharger speed. As commented by Baines et al. (1994) the energy content under pulsating flow conditions could be higher than under steady conditions and it is not necessarily that the turbine efficiency is increased. Another important factor, which is difficult to take into
account, is the heat transfer from the turbine to the compressor that affects the measurements of the temperature rise of the compressed air.

Baines & Yeo (1991) showed that the flow variations measured under pulsating flow at the rotor inlet were similar to those measured under steady flow (for partial admission conditions, measurements performed on a twin-entry turbine) and that no pulsating effects could be seen at the rotor exit. The conclusion was that large scale energy transfer in the rotor eliminates pulsating flow effects, which imply that the rotor can be seen as a quasi-steady flow device. This would imply that the dynamic effects occur in the volume upstream the rotor. This was at the time contradicted by Winterbone & Alexander (1990) and Winterbone et al. (1991) who measured the pressure at the rotor inlet at different azimuth angles with static pressure taps on the volute wall. They concluded that the dynamic effects in a turbine must come from the rotor due to that the pressures at the rotor inlet for a pulsating flow showed similar behaviour as for steady flow.

There are several difficulties that have to be overcome when it comes to compute the turbine performance at pulsating inflow conditions from experimental data. First of all there is a phase shift between the measured isentropic conditions upstream the turbine and the extracted work measured at the shaft. The flow approaches the rotor at different azimuth angles around the volute, which makes it difficult to find a fixed parameter to compensate for the phase shift. During a pulse event the convection speed or the bulk velocity changes significantly as well as the local temperature, which influences the speed of sound, and has to be considered. As shown by Hellström & Fuchs (2007) there is also a phase shift between the mass flow and the pressure of up to 52° which changes during a pulse event. Ehrlich & Lawless (1997) performed extensive measurements of the flow characteristics on a 6-cylinder medium size diesel engine to analyse the on engine turbine performance, i.e. for a pulsating flow including temperature fluctuations. Pressures were instantaneously measured before and after the turbine with strain gauge pressure transducers, temperatures with frequency response compensated fine wire thermocouples and flow velocities via Particle Image Velocimetry (PIV) to compute the instantaneous mass flow rate. The measurements showed that the flow velocity, pressure and temperature within the volume of the exhaust manifold and turbine propagates with different speeds. This suggests that a common propagation velocity for the phase shift of the physical parameters used in the flow rig experiments to compute the instantaneous turbine efficiency might not be sufficient. The measurements also showed that there are significant compressibility effects in the turbine, which implies that the turbine performance under pulsating flow conditions departs from a quasi-steady turbine performance description. The velocity measurements showed that the flow remains relatively uniform during the initial opening of the exhaust valve, but as the axial velocity decreases cross sectional velocity components can be found. Lam et al. (2002) performed
a computational study on the instantaneous performance of an entire turbine stage for unsteady flow conditions using the moving reference frame method to account for the rotation of the wheel, which was justified with the motivation that the pulse frequency is 40 times lower than the rotation of the rotor. The instantaneous turbine efficiency was computed for the entire stage and solely for the rotor. The rotor was shown to act as a quasi-steady flow device while the entire stage deviated from the quasi-steady behaviour. This implies that the volute influences the unsteady flow conditions seen by the rotor. Hellström & Fuchs (2008) who performed LES using the sliding mesh technique to model the rotation of the wheel concluded that the entire stage behaves in an unsteady manner due to inertia of the system and a non-constant phase shift between the mass flow, pressure and the shaft power output.

Researchers have used different phase shifting techniques to correlate these measures. The assumptions have been that the flow hits the rotor at a single point and that the fluid travels by the bulk and/or sonic velocity to compute the time difference from the measurement locations. Dale & Watson (1986) used the sonic velocity while Winterbone et al. (1991) and Baines et al. (1994) used the velocity of the bulk flow for phase shifting. Arcoumanis et al. (1999) derived instantaneous efficiency for a mixed-flow turbine, which shows a similar behaviour as for the radial turbine, comparing the two methods based on different velocities for phase shifting. Results showed that the method used for phase shifting gives a significant effect on the derived instantaneous efficiency. Szymko (2006) and Rajoo (2007) used the sum of the bulk and sonic flow velocity for deriving the instantaneous efficiency for mixed-flow turbines on a pulsating flow rig due to that it gave a good correlation between the pressures measured at different locations of the turbine. However, Rajoo (2007) questions the evaluation of the turbine performance under unsteady flow condition via instantaneous efficiency due to the need of phase shifting.

Lujan et al. (2001) tried a somewhat different approach to derive the on-engine turbine efficiency. Instead of measuring all the thermodynamic properties they used a calibrated 1D gas-dynamic engine model to extract some of the parameters in order to derive the turbine efficiency. They calculated a cycle-average turbine efficiency, which was 75% of the efficiency corresponding to the values from the turbine manufacturer measured under steady flow. However, the measurements did not include turbocharger speed fluctuations. Westin (2005) used a similar approach to determine the thermodynamic properties and to compute the on-engine turbine efficiency. The efficiency calculations did not include any phase shift between measured thermodynamic properties upstream and downstream the turbine and the turbocharger shaft acceleration since no significant phase difference could be seen.
CHAPTER 3

Turbine Models for Pulsating Flow Conditions

Several types of models have been suggested to model the performance of IC-engine components. The main types of models are mean-value models, empirical models, emptying and filling models, wave-action models also called one-dimensional models and multi-dimensional models. The review will primarily consider turbine models for the intention to be used within engine simulations. Full 3D turbine computations have not been included in the review, even though these models have proven to give satisfactory results, since they are today not suited for routine engine simulations in the industrial design process, due to the CPU-time required.

The easiest way to model the turbine is to use tabulated measured turbine performance data from a steady flow rig, with the assumption that the turbine works as a quasi-steady flow device as suggested by Benson (1982). The turbine is modelled as a unit, setting a boundary condition for the manifold. This approach neglects the volume of the turbine and its length to account for wave propagation or compressibility effects. However, as has been shown by several researchers, see Section 2.5, these effects can not be neglected for pulsating flow conditions, which the turbine models suited for pulsating flow conditions tries to account for.

3.1. Filling and Emptying Models

Filling and emptying models, or in other words zero-dimensional models, model the volume of the geometry, but does not consider any spatial dimensions such as the length of the flow path. They assume that the flow parameters are functions of time only and not space, and consider compressibility effects, but not wave propagation. The compressibility effects are modelled through an ODE integrated in time.

Bhinder & Gulati (1978) applied a filling and emptying model for a nozzle-less radial turbine first presented by Kastner & Bhinder (1975) to a pulsating flow case. It was claimed that the filling and emptying approach is justified due to that the length of the exhaust manifolds and the flow path in the volute are relatively small and the time for wave propagation can therefore be neglected. The work output was based on the Euler turbo machinery equation, Equation 2.4, together with an incidence loss model to account for
losses in the rotor based on the review regarding losses in a radial turbine by Benson (1970). Another argument for using a filling and emptying model was presented by Baines et al. (1994) who argue that a wave action model is excessive since the pulse frequency is usually two orders of magnitude smaller than the characteristics of the turbine rotor process. The process can therefore be seen as a continuous change in the operational point of the turbine, i.e. as a quasi-steady process. Baines et al. (1994) concluded from earlier measurements that the rotor works as a quasi-steady flow device and can therefore be modelled accordingly. The model presented is a filling and emptying model of a single entry turbine consisting of a volume representing the volute for mass accumulation and a boundary representing the rotor with losses computed from empirical relations. Comparison with measurements showed that the efficiency is strongly affected by the phasing of the mass flow trace, the pressure and the output torque. The "hysteresis loops" considered, were most closed when the assumption that the fluid transport is dominated by the convection speed rather than the propagation speed of the pressure waves, i.e. the speed of sound superimposed on the flow velocity. The results with respect to total-to-static efficiency over $U/C$ were in general consistent with measured data, exhibiting a similar looping behaviour over similar ranges of efficiency and velocity ratio, Figure 3.1.

Another approach to model the turbine is to assume that the turbine works as a nozzle, as presented by Watson & Janota (1982), with an effective area corresponding to the pressure drop for a specific mass flow rate across it. The problem here is when choking occurs, which is not unusual for a turbine in automotive applications. A nozzle chokes at a pressure ratio of approximately 1.9 while choking actually occurs at a pressure ratio of approximately 3 for a radial turbine, see e.g. Figure 4.1. To come around the problem Watson and Janota hinted that there is a possibility to use two nozzles in series. This
approach was tested by Payri et al. (1991) who proposed a filling and emptying model consisting of two nozzles in series to model the stator and the rotor with an intermediate volume to account for compressibility effects. The model was extended to model twin-entry turbines by Payri et al. (1996) and to variable geometry turbines by Luján et al. (2006) and Serrano et al. (2008). The models were used with the assumption that the turbine works as a quasi-steady flow device, that the inflow to the turbine is radial and that there is no swirl at the exit of the rotor at the design point of the turbine, i.e. at maximum efficiency. The second and third assumption implies a reaction degree of 0.5 as shown by Watson & Janota (1982).

The effective nozzle areas were derived with respect to the reaction degree at the design point from stationary measured turbine performance data. As pointed out by Payri et al. (1996), the reaction degree will change during a pulse event or for different operating points in use. Payri et al. (1996) did also show results from varying the effective area of the nozzles by an increase and a decrease by a factor of two. The results for the different effective nozzle areas did not show any significant differences for the pressure traces upstream and downstream the turbine. However, Serrano et al. (2008) compared pressure traces to measurements both in the time and the frequency domain with promising results.

3.2. One-Dimensional Models

The 1D modelling approach, which assumes that the flow is essentially one-directional, is often assumed to be adequate for turbine modelling, as pointed out by Lawless (1997). However, the 1D approach neglects secondary flow effects such as swirl, a skewed velocity profile etc. and it will not take secondary flow effects into account as shown by Renberg et al. (2009). Hellström & Fuchs (2008) showed that secondary flow effects from e.g. different inlet flow conditions can have a significant effect on the turbine power output due to varying incidence angles onto the turbine blades. However, several researchers have tried to come up with a 1D model to predict the turbine performance under pulsating flow conditions.

Chen & Winterbone (1990) estimated the Strouhal number for the volute and rotor separately, which showed that dynamic effects should be considered for the volute but not for the rotor. Chen & Winterbone (1990) presented a 1D model for a mixed flow turbine where the flow in the volute was modelled as a tapered pipe to replicate the cross-sectional area of the volute. The volute was divided into three stages. The first stage was defined from the turbine entry to the volute tongue. At the second stage, from the volute tongue, the fluid was assumed to accelerate due to the tapered pipe. At the third stage the flow was assumed to reach the rotor at an azimuth angle of 180 degrees. The exit area of the tapered pipe was derived according to,
\[ A_2 = \frac{2\pi}{(2\pi - \phi_2)} A_{2,\text{real}} \] (3.1)

which approximately can be assumed to be the area at the average distance propagated through the turbine housing for all gas particles, where \( \phi_2 \) is the set azimuth angle and \( A_{2,\text{real}} \) is the actual cross-sectional area. The rotor was modelled as a quasi-steady flow device since the estimated Strouhal number for the rotor was small. The losses in the rotor were implemented as a source term in the momentum equation based on a frictional loss coefficient,

\[ -\frac{C}{4A} C_f p u^2 \text{sgn}(u) \] (3.2)

where \( C \) denotes the wetted perimeter at each cross section. The loss coefficient was based on existing NASA models, similar to those used in Rital, see Section 4.2.1. The loss term was calibrated at the design point and then run at off-design operation for validation at different steady flow conditions. The model was improved by Chen et al. (1996) with respect to the loss modelling, which were divided into several parts as incidence losses, flow losses due to tip clearance etc. The loss coefficients were again calibrated with measured steady flow data and validated under pulsating flow conditions for different pulse frequencies and amplitudes. The results showed that the pulse amplitude has a larger effect on mass flow rate than the pulse frequency. Thus, the turbine swallowing capacity increases under pulsating flow conditions, which implies that a quasi-steady approach will underestimate its flow capacity. Chen et al. (1996) concluded that a true model for the rotor, which consider flow dynamics and can take the circumferential flow variations at the rotor inlet into account is necessary, even if the Strouhal number for a single flow passage is relatively small.

Miyauchi & Yoshiki (1994) also developed a quasi-1D model for a nozzled radial turbine from geometrical data. The flow passage within the volute was modelled as a tapered pipe with the length of the volute to an azimuth angle of around 180 degrees. The model was based on the equations governing the flow in 1D including area changes. Losses were modelled as source terms in the form of body and friction forces calibrated at steady flow conditions. Results for an error of the time averaged mass flow rate was shown, which shows an overestimation by up to 20\% for a relative velocity amplitude of 0.9 compared to measurements. Effects from pulse frequency is shown but not commented since the pulse frequency had a lesser effect than the pulse amplitude. A review was presented in the subject by Winterbone & Pearson (1998). The conclusion was that there is no turbine model actually operating under pulsating flow conditions, even though introducing lengths and volumes into a quasi-steady model can result in a more accurate turbine model.
Another model for the turbine was developed by Hu & Lawless (2001) to model the turbine performance under pulsating flow conditions which relies on the experimental results presented by Ehrlich & Lawless (1997) from on-engine measurements. The model was developed for a vane-less radial twin-entry turbine with the assumption that the secondary flow components are negligible and that the turbine is adiabatic. The model is based on the quasi 1D Euler equations with source terms,

$$\dot{W} + F_x = C$$

(3.3)

where $\dot{W}$ is the vector of conservative variables, $F$ the flux function of $\dot{W}$ and $C$ denotes the source terms to account for body and surface forces and mass flow through the boundaries to account for different length of the flow path,

$$\dot{W} = \begin{bmatrix} \rho \\ \rho u \\ \rho \varepsilon \\ \rho \varepsilon + p \\ u(\rho \varepsilon + p) \end{bmatrix}, \quad F = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ 0 \end{bmatrix}$$

$$C = \begin{bmatrix} -\rho u \frac{dA}{dx} - \dot{m}_{\text{out}} A \\ -\rho u \frac{dA}{dx} + F_{\text{n}} + F_{\text{B}} - \frac{(\dot{m}u)_{\text{out}}}{A} A \\ -\rho u \frac{dA}{dx} + F_{\text{n}} - \frac{(\dot{m}H)_{\text{out}}}{A} A \\ -\rho u \frac{dA}{dx} + F_{\text{n}} + F_{\text{B}} - \frac{(\dot{m}H)_{\text{out}}}{A} A \end{bmatrix}$$

(3.4)

$\rho$ denotes density, $u$ gas velocity, $\varepsilon$ specific internal energy, $H$ total enthalpy, $p$ pressure, $\dot{m}$ mass flow rate and $A$ denotes the cross-sectional area. From thermodynamics, $p = (\gamma - 1)(\rho e - 0.5pu^2)$ which completes the set of equations. $F_S$ denotes the x-component of the shear forces per unit length and $F_B$ the centrifugal body forces per unit length,

$$F_B = \begin{cases} \frac{C_F \omega^2 \rho A}{R} & \text{(Radial Section)} \\ 0 & \text{(Axial Section)} \end{cases}$$

(3.5)

The losses were only confined to the rotor from where the majority of the losses in a turbine originate. The loss terms for the shear and body forces were calibrated with stationary performance data where $F_B$ considers the rotation of the rotor. Nozzles were incorporated and set to a certain inflow angle towards the rotor to model the inflow into the rotor channels, since a varying inlet flow angle could not be incorporated into the 1D model. The rotor was divided into a radial and an axial section since the power output of the rotor was assumed to only come from its radial section. The second source term models the mass flow rate into the nozzles at different azimuth angles. The stagnation pressure and temperature along the volute are given to the individual nozzle channels as the inlet stagnation pressure and temperature.

An instantaneous efficiency of the turbine was computed from upstream and downstream conditions as the ratio between the actual power output and the power output for an isentropic turbine assuming a constant specific heat capacity,
\[ \eta_{TS} = \frac{\int (\dot{m}_{in} T_{03} - \dot{m}_{out} T_{04}) dt}{\int \dot{m}_{in} (T_{03} - T_{is}) dt} \]  

(3.6)

where the mass flow rates and temperatures for the actual power output are taken upstream and downstream the turbine, with \( T_{03} \) the total temperature upstream the turbine and \( T_{is} \) the isentropic static temperature downstream the turbine, i.e. expansion over the turbine without losses. Mass flow rate and isentropic temperature to compute the ideal work output, were taken from steady flow performance measurements of the turbine. The model was refined by King (2002) whose model consider choking and has a more refined numerical solver than the method of characteristics previously used. King (2002) validated the model with the measured data from Ehrlich & Lawless (1997), which followed the trends with respect to swallowing capacity and ideal output power. However, actual power output could not be validated since there were no measurement data on the rotational speed of the turbocharger shaft to derive the shaft torque from. The significant differences between ideal and actual power output were referred to as mass accumulation by Ehrlich, i.e. compressibility effects, but King (2002) points out that this has rather to do with large torque variations over the pulse cycle.

Costall (2007) presented a turbine model, also based on the quasi-1D Euler equations with an added pressure loss coefficient to account for losses. The geometry of the turbine was modelled as two pipes in series representing the volumes and the flow paths lengths of the stator and rotor, respectively. The length of the pipe representing the volute was set to the geometrical centreline of the volute assuming that the flow reaches the volute at 180 degrees of azimuth angle from the volute tongue. The diameter of the pipe was chosen to give a correct volume of the volute including the volume of the pipe from the measurement plane to the volute. The second pipe represented the volume of the rotor. An adiabatic pressure loss boundary in the form of a lookup table from stationary flow rig measurements was added between the pipes representing the volute and the rotor to account for pressure losses within the turbine. The inlet boundary was implemented as a transmissive boundary, which is non-reflective and active in order to reproduce a given mass flow trace from the measurements. In comparison to the model presented by Hu & Lawless (2001), the model developed by Costall (2007) does not consider the rotation of the turbine wheel, which influences the body forces, nor the fact that the mass flow rate along the volute diminishes. The model was compared visually with data from measurements with a pulse frequency of 20 to 80 Hz. The results are in good agreement for the 20 Hz case, but the agreement declines for the 40 Hz case, as shown in Figure 3.2, and for higher frequencies.

Costall et al. (2009) extended the model to predict the output power of the turbine. The power output was computed from the predicted instantaneous isentropic power output at the rotor boundary multiplied by a stationary
3.2. ONE-DIMENSIONAL MODELS

Efficiency coefficient from stationary turbine performance measurements. The comparison of the predicted power output to measurement in the time domain deviated as much as 50% at a part of the pulse. Unfortunately, the errors were not really quantified.

Piscaglia et al. (2007) used a similar approach where the turbine was modelled with a volume to account for compressibility effects connected to a boundary condition accounting for the pressure losses in the rotor by a pipe to consider wave propagation. However, the model did not capture the hysteresis effects that could be found in the flow rig measurements. Another quasi-1D turbine model was presented by Macek & Vítěk (2008), who modelled the flow through the turbine components along an assumed streamline to consider pulsating flow conditions. However, no results were presented.
CHAPTER 4

1D Engine Simulation Methods

The quasi-1D engine simulation software GT-Power was used for the engine simulations conducted in this work, Gamma Technologies (2004), with the engine configurations according to Figure 4.3 and 4.4. To model an internal combustion engine, a model is setup from its different components as e.g. cylinders, valves, turbocharger components, flow splits and straight and bent pipes. Special objects exist as e.g. inlet or outlet boundaries, orifice connections etc., which are used as boundaries between the objects or for the whole domain.

The quasi-1D model, Equation 4.1 - 4.3, is based on the equations governing the flow, i.e. the mass, momentum and energy conservation equations, with the assumption that the flow is uniform across the flow direction. Cross-sectional area-changes are still considered. Comparing the 1D flow equations used for the engine simulations to the governing equations in 3D, Equation 5.1 to 5.3, shows that influence from span wise velocity components onto the axial velocity component are neglected. A source term is added to the momentum equation to account for pressure losses and the influence from span wise velocity components onto the axial velocity component. The source term is based on tabulated data from measurements. A second source term is added to account for frictional losses due to surface roughness. The equations are used for the flow in pipe like objects as straight and bent pipes and for flow splits in bifurcating/merging pipes,

\[
\frac{\partial (pA)}{\partial t} + \frac{\partial (\rho uA)}{\partial x} = 0
\] (4.1)

\[
\frac{\partial (\rho uA)}{\partial t} + \frac{\partial (\rho u^2A)}{\partial x} = -\frac{\partial (pA)}{\partial x} - C_p \frac{\rho u^2A}{2\Delta x} - C_f \frac{\rho u^2 A}{2D}
\] (4.2)

\[
\frac{\partial (e_0 \rho A)}{\partial t} + \frac{\partial \left( \rho u \left( e_0 + \frac{p}{\rho} \right) A \right)}{\partial x} - \frac{\partial q}{\partial x} = 0
\] (4.3)

\(\rho\) denotes density, \(T\) temperature, \(p\) pressure, \(u\) flow velocity, \(e_0\) total specific energy and \(A\) the cross-sectional area. The source terms in the momentum equation, Equation 4.2 comprises the skin friction coefficient (\(C_f\)) and the pressure loss coefficient (\(C_p\)), which can be adjusted by the user. To close the
4.2. TURBINE MODELLING

set of equations, the assumption of an ideal gas is used to get the equation of state,

$$ p = \rho RT $$

(4.4)

Sub models are used for engine components where 3D flow phenomena occur, such as the turbocharger components, the valves or where chemical reactions take place as the in-cylinder process. For the engine simulations, the sub models of the flow components as the turbine, compressor and the valves, were based on steady flow rig measurements from the manufacturers. The heat release rate for the in-cylinder process was measured for the specific engine and the specific operational points simulated.

The two different turbine modelling approaches used, based on measurements, are described in Section 4.2.

4.1. Numerical Methods

The discretized set of governing equations is solved numerically for the pipe objects, which are divided into sub volumes. The spatial discretization is based on a first order upwind scheme. The flow solution is obtained through an explicit time integration of the governing equations by a first order Euler scheme. The explicit solver is recommended in order to capture flow pulsations, even though the time stepping has to be shorter than for the implicit solver to meet the recommended CFL-requirement,

$$ \frac{\Delta t}{\Delta x} (|u| + a) \leq 0.8 $$

(4.5)

where $a$ denotes the speed of sound.

4.2. Turbine Modelling

A standard approach to characterize the turbocharger performance is via turbine and compressor maps. These maps are based on measured performance data from a steady flow rig and used as lookup tables within the engine simulation code. The turbocharger performance data from the measurements are specified in terms of mass flow rate and efficiency versus rotational speed and pressure ratio, Figure 4.1. To consider inlet conditions, i.e. for different inlet temperatures and pressures, the mass flow rate and rotational speed are specified as reduced or corrected according to Equation 4.6 - 4.9. The corrected values for mass flow and rotational speed are defined to give the same axial and blade Mach numbers, respectively, as in the measurements. The pressure ratio is defined as total to static according to Equation 4.10.
\[ N_{\text{red}} = \frac{N}{\sqrt{T_{0,\text{in}}}} \] (4.6)

\[ N_{\text{corr}} = \frac{N}{\sqrt{T_{\text{ref}}}} \] (4.7)

\[ \dot{m}_{\text{red}} = \frac{\dot{m}}{p_{0,\text{in}}} \sqrt{T_{0,\text{in}}} \] (4.8)

\[ \dot{m}_{\text{corr}} = \frac{\dot{m}}{p_{0,\text{in}}} \sqrt{T_{0,\text{in}}} \sqrt{T_{\text{ref}} \frac{p_{\text{ref}}}{p_{\text{out}}}} \] (4.9)

\[ PR = \frac{p_{0,\text{in}}}{p_{\text{out}}} \] (4.10)

**Figure 4.1.** Measured flow characteristics and efficiency for a typical radial turbine on a steady-flow flow rig. Each line correspond to a constant speed.

For use within the engine simulation software the data set has to be interpolated and extrapolated. The interpolation and extrapolation routine used for the turbine, is performed through a curve fit where the measured performance data is assumed to collapse on a single curve, Figure 4.2. The compressor data is mainly interpolated except at low speeds where the data is extrapolated. This is done as a linear interpolation down to a specified lowest value.

The engine simulation code uses the turbine and compressor sub models to evaluate the mass flow rate and efficiency outputs for a given rotor speed and pressure ratio at every time step during the simulation. The power produced/consumed by the turbine/compressor are then computed from the change in enthalpy over the device together with their respective efficiencies.
4.2. TURBINE MODELLING

Figure 4.2. Normalized turbine performance data.

from the measured performance maps, where the pressure ratios are calculated from the adjacent volumes down- and upstream of the turbine/compressor,

\[
P_T = \eta_T \dot{m} c_p T_{0,in} \left(1 - \left(\frac{p_{out}}{p_{0,in}}\right)^{1-\gamma/\gamma}\right) \tag{4.11}
\]

\[
P_C = \eta_C \dot{m} c_p T_{0,in} \left(1 - \left(\frac{p_{out}}{p_{0,in}}\right)^{\gamma-1/\gamma}\right) \tag{4.12}
\]

The turbocharger speed is derived from the torque imbalance associated with the compressor and turbine power between the turbine, compressor and frictional losses in the shaft assembly according to,

\[
\frac{d\omega_{TC}}{dt} = \eta_{mech} P_T - P_C \tag{4.13}
\]

To model a VGT, different turbine performance maps are used for different rack positions and then interpolated in between. A twin-entry turbine is modelled with two turbine objects based on a single volute turbine map connected in parallel since the turbine performance data is measured at equal admission, i.e. at equal flow rates between the two inlets. By default the mass flow to each turbine or turbine inlet is divided evenly between the two. To reproduce the pressure trace upstream the twin-entry turbine an orifice connection has to be applied between the two inlet pipes to the turbine volute to allow for cross flows within the turbine as shown by Winterbone & Pearson (1999). Unfortunately,
4. 1D ENGINE SIMULATION METHODS

no model to compute the area of the orifice connection has been found in the literature and has to be set by the user.

Several problems arise when using turbine performance data to model the turbine performance,

1. Performance data for low powers are usually not measured, due to difficulties to obtain data with high accuracy, and therefore have to be extrapolated.
2. The quasi-steady assumption will lead to that the phase shift between the inlet and output power will not be considered, nor compressibility effects within the volute and rotor.
3. For turbines with twin-entry volutes the performance is usually measured under equal admission conditions, which gives erroneous data for partial admission conditions.
4. Heat losses are not considered in the performance maps.

To compensate for the volume of the volute and the rotor, two pipes were added to the turbine sub model. One pipe upstream the sub model and one pipe downstream with the same diameter as the engine manifolds and a length to give the correct volume of the turbine volute and rotor, respectively. The reason for the added pipes was to account for compressibility effects to consider wave propagation, even though the pipes did not have the correct lengths for the wave propagation.

4.2.1. Mean-Line Turbine Modelling

The mean-line radial turbine design tool Rital, developed by Concepts NREC (2005), was used to extend the existing measured turbine performance data for the engine simulation. Rital is based on semi-empirical models described by Baines (1998) and in the textbook by Moustapha et al. (2003). Rital calculates the expansion process, velocities and flow angles throughout the different stages of the turbine for any specified geometry and inlet condition. The software requires turbine geometry data such as area and radius ratio, wheel inlet, wheel exit, hub and tip diameter, blade angles etc. For prediction of aerodynamic losses, several parameters, which are difficult to measure, have to be given, such as optimum incidence angle and rotor exit deviation angle. The software utilizes loss coefficients for a range of possible internal flow mechanisms that has to be set by the user. The components that Rital supports are the volute, nozzles with vanes, rotor and diffuser. However, the only components described here are the volute and rotor since they were the only ones needed for the turbine modelled.

The choice of the area to radius ratio, i.e. the inlet cross sectional area and the radius from the turbine axle to the geometrical centreline of the volute at the position of the volute tongue, determines the exit flow angle from the volute, which has a large influence on the swallowing capacity of the turbine. The area
4.2. TURBINE MODELLING

to radius ratio is assumed to be a linear function with respect to the azimuth angle to obtain a uniform mass flow distribution around the volute. Hence, the volute model in Rital is solely based on the area to radius ratio assuming an incompressible flow. The losses in the volute are modelled as total pressure losses, frictional losses and blockage representing losses due to boundary layer growth and secondary flow, which cannot be measured separately.

The inlet angle of the flow relative to the inlet blade angle of the rotor, i.e. the incidence angle is computed so that the radial velocity components satisfy the continuity equation for the incoming flow to the volute. At a non-optimal incidence angle work has to be put in to turn the flow according to the blade passage. This loss is called the incidence loss and is modelled as,

\[ L_i = \frac{1}{2} W_2^2 \sin^2 (\beta_2 - \beta_{2,\text{opt}}) \]  \hspace{1cm} (4.14)

based on the optimal incidence angle. \( W_2 \) denotes the relative flow velocity with respect to the blade tip speed according to Figure 2.7.

Main losses within the rotor from the leading edge to the throat are modelled as passage losses. Passage losses consists of losses due to secondary flow, blockage and loss of kinetic energy due to growth of boundary layers,

\[ L_p = K_p \left[ \left( \frac{L_H}{D_H} \right) + 0.68 \left[ 1 - \left( \frac{r_4}{r_2} \right)^2 \left( \frac{\cos \beta_m}{b_3/c} \right) \right] \right] \frac{1}{2} (W_2^2 + W_3^2) \]  \hspace{1cm} (4.15)

where \( K_p \) is a total pressure loss coefficient, \( L_H \) and \( D_H \) the passage hydraulic length and diameter, respectively. \( b_3 \) denotes the exit blade height and \( c \) the blade chord. Then there are trailing edge losses influenced by the rotor exit deviation angle defined as the difference between the exit blade angle and the angle of the flow leaving the rotor. The deviation angle affects the velocity field distribution downstream of the rotor. Other losses are; clearance losses, which are affected by the clearance between the tip of the rotor and the turbine housing and the clearance at the back plate of the rotor and windage losses. The rotor is modelled in two parts with a radial- and an axial part. The clearance losses are proportional to the absolute velocities at each respective direction.

Unknown parameters such as loss coefficients, optimum incidence and deviation angle are required and have to be set. These parameters cannot be measured directly and therefore the model has to be calibrated against measured turbine performance data. However, even though the maps can be extended with the method described the modelling of the turbine performance is still performed with the assumption that the turbine works as a quasi-steady flow device.
4.3. Experimental Setup

The engine simulations are validated with measurements performed at the division of Internal Combustion Engines, KTH. The main parameters measured on the engine are pressures and temperatures along the gas exchange system, mass flow rate of air, cylinder pressure and turbocharger speed. Measurements were performed on a heavy-duty diesel engine with a single- and a two-stage turbocharger system shown in Figure 4.3 and 4.4. For the development of the two-stage turbocharger system including a short-route for EGR, a pre-simulation of the engine was performed as a help to choose configuration and reasonable sizes of the turbocharger units. The main prerequisite was to achieve high turbocharger efficiencies with an EGR rate of 30% at 1000 to 2200 rpm at high loads, i.e. with a maximum cylinder pressure limit of 220 bars and a minimum air-to-fuel corresponding to $\lambda > 1.2$. The two-stage turbocharger system chosen, consists of a twin-entry radial turbine at the high pressure (HP) stage to prevent exhaust pulses from different cylinders to interfere and backflow of exhaust gases into the cylinders, a VGT at the low pressure (LP) stage to control the back pressure and the pressure drop balance over the stages, using a sliding nozzle-ring mechanism controlled via an electric motor. The sizes of the units are given in table 1, as critical area for the turbines, i.e. the area at the volute tongue and as impeller exducer tip diameter for the compressors in relation to the units used on the engine, originally. A picture of the gas exchange system of the rebuilt engine installed in the test cell is shown in Figure 4.5.

<table>
<thead>
<tr>
<th>Turbocharger unit</th>
<th>Size relation</th>
</tr>
</thead>
<tbody>
<tr>
<td>HP Compressor</td>
<td>100%</td>
</tr>
<tr>
<td>LP Compressor</td>
<td>110%</td>
</tr>
<tr>
<td>HP Turbine</td>
<td>40%</td>
</tr>
<tr>
<td>LP Turbine</td>
<td>160%</td>
</tr>
</tbody>
</table>

For the single-stage turbocharged engine, pressures and temperatures were measured at the following positions, range of pressure sensor and size of thermocouples given in parenthesis, respectively,

- before the compressor, (abs. 1 bar, 3 mm)
- after the compressor, (rel. 4 bar, 3 mm)
- before the intake plenum, (rel. 4 bar, 3 mm)
- before the turbine at each volute entry, (rel. 6 bar, 3 mm)
- after the turbine, (rel. 1 bar, 3 mm)
4.3. EXPERIMENTAL SETUP

For the two-stage turbocharged engine, pressures and temperatures were measured at the following positions,

- before the LP-compressor, (abs. 1 bar, 3 mm)
- after the LP-compressor, (rel. 4 bar, 0.5 and 3 mm)
- before the HP-compressor, (rel. 6 bar, 0.5 and 3 mm)
- after the HP-compressor, (rel. 10 bar, 0.5 and 3 mm)
- before the intake plenum, (rel. 10 bar, 0.5 and 3 mm)
- before the HP-turbine at each volute entry, (rel. 10 bar, 1.5 and 3 mm)
- after each EGR-cooler, (rel. 10 bar, 1.5 and 3 mm)
- after the HP-turbine, (rel. 6 bar, 1.5 and 3 mm)
- after the LP-turbine, (rel. 1 bar, 1.5 and 3 mm)

The pressure measurements at the exhaust side were performed with two sets of sensors. The first set mounted close to the gas via water-cooled blocks to measure the pressure fluctuations. The second set was mounted on 0.5 to 1.0 m long tubes to prevent temperature drift when measuring cycle-averaged pressures during transient operation.

The amount of EGR was measured via the ratio of the $CO_2$ level added to the amount of fresh air in the intake before the plenum and after the LP-turbine,

$$EGR = \frac{([CO_2]_{intake} - [CO_2]_{air})}{[CO_2]_{exhaust}} \quad (4.16)$$

Due to the slow response time of the $CO_2$ analyzers, the EGR measurements could only be performed during steady-state operation of the engine.

For both engine configurations the mass flow rate air was measured before the compressor. Cylinder pressure was measured at cylinder 6 only. The injected amount of fuel was measured by a fuel weighting scale during steady-state operation. For transient operation the injected amount of fuel was obtained from the duration of the needle lift of the PDE injector, related to the fuel flow at steady-state operation according to the fuel weighting scale.

The CELL4 data acquisition and test bed control system was used for the measurements, built in-house. The data acquisition system enables both high and low frequency signals to be measured, i.e. on a cycle-averaged or on a crank-angle resolved basis. Crank-angle resolved measurements were sampled with 0.1° crank-angle resolution with a 12-bit A/D-converter. PIC micro controllers measured digital input, i.e. crank-angle encoder pulses. The PIC micro controllers additionally produced single engine revolution averages for analogue inputs, accomplished by buffering transducer input from external 12 bit A/D converters at 10 kHz and averaging the buffered data once every engine revolution. Analogue low-pass filters, chosen with regard to sampling
frequency and transducer natural frequency to avoid aliasing, and digital IIR-filters with zero phase distortion were used to assure undisturbed data with low noise.

For pressure measurements piezo electric pressure transducers from Kistler and strain gauge transducers with a steel diaphragm from Gems were used. The pressure transducers from Kistler were used for the crank-angle resolved measurements at the exhaust side. The Gems transducers were used for the intake side and for cycle-averaged measurements at the exhaust side. Static calibration of the pressure transducers were accomplished by a calibrated pressure indicator and a hand pump. The transducers were calibrated with an error below 0.5 % and 0.08 % full scale for the transducers from GEMS and Kistler, respectively.

Gas temperatures were measured with shielded 0.5 to 3 mm K-type thermocouples from Pentronic. During transient operation the 0.5 mm thermocouples were used at the intake side to try to follow the increase in temperature during engine transients. 1.5 mm thermocouples were used at the exhaust side due to the risk of breakage, using the thin ones, and destroying the turbine.

Inlet air mass flow rate was measured with a Hot-film meter, Bosch HFM5 mounted in a pipe with a larger area compared to the pipe used by the manufacturer to extend the measurement range. Calibration with the air flow rig at AVL in Jordbro, Sweden showed an error below 1 % according to the manufacturer’s specifications after scaling with the larger pipe area.

A piezo electric water cooled pressure transducer from AVL was used for the cylinder pressure measurements. A dead weight scale was used for the cylinder pressure transducer calibration. The linearity error up to 250 bar was found to be within 0.5 %.

Turbocharger speeds were measured by measuring and storing the times of the blade passages according to a clock with a counting frequency of 10 MHz. The blade passages were detected with an eddy-current probe from Micro-Epsilon mounted in the compressor housing.

The CO₂ levels to compute the amount of EGR for the two-stage turbocharged engine was measured with two non-dispersive infra red light analyzers from Maihak, Boo-Instruments.
Figure 4.3. Six-cylinder engine with a single-stage turbocharger system.
Figure 4.4. Six-cylinder engine with two-stage in series turbocharging system, two-stage intercooling and short route EGR. The arrow denotes the VGT.

Figure 4.5. Six-cylinder 12 litre engine with two-stage in series turbocharging system, two-stage intercooling and short route EGR installed in the test cell at KTH.
CHAPTER 5

Methods for Reduced Models Based on LES

Two reduction methods of the governing equations studied within this work with the aim to improve engine simulation tools will be described in the following. The governing equations are reduced in a systematic way based on data from Large Eddy simulations (LES). Data for a double bent pipe, a radial turbine and a circular duct with an orifice plate are used to demonstrate and apply these approaches for the model reduction.

5.1. Governing Equations

The characteristic length- and time-scales for fluid flows are significantly larger than the discrete structure of the fluid considered. It is therefore a good approximation to describe the physical properties of fluids through continuous functions in space and time. The governing equations describe the conservation of mass, momentum and energy and can be written according to Equation 5.1, 5.2 and 5.3, respectively. For completeness, the energy equation is complemented with the equation of state, Equation 5.4, assuming an ideal gas.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \quad (5.1)
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (5.2)
\]

\[
\frac{\partial (\rho e_0)}{\partial t} + \frac{\partial (\rho u_j e_0)}{\partial x_j} = -\frac{\partial (p u_j)}{\partial x_j} + \frac{\partial (u_i \tau_{ij})}{\partial x_j} - \frac{\partial q_j}{\partial x_j} \quad (5.3)
\]

\[
p = \rho RT \quad (5.4)
\]

where \( q_j \) denotes the heat flux. The viscous shear stress tensor \( \tau_{ij} \) represents the stress from fluid motion. The stress tensor for a Newtonian fluid, which is a fluid whose stress is linear to its strain rate, i.e. the gradient of the flow velocity perpendicular to the direction of the shear, is defined as,

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \quad (5.5)
\]
This system of equations, is a set of non-linear PDEs with a mixed hyperbolic-parabolic character.

5.2. Computations of Turbulent Flows

The flows that we are dealing with in most engineering applications are turbulent by nature. Such flows are characterized by large Reynolds numbers \( Re \), which is a dimensionless number expressing the ratio between inertial and viscous forces,

\[
Re = \frac{U \cdot l}{\nu}
\]  

where \( U \) is a characteristic velocity, \( l \) a characteristic length and \( \nu \) is the kinematic viscosity. Turbulence is a chaotic state of the flow consisting of motions in the form of three-dimensional unsteady eddies at different sizes. The large-scale eddies are the most energetic ones and are dependent on the geometry itself and the flow conditions near the boundaries. The larger eddies deform through vortex stretching and transfer their energy into smaller eddies. This is an on-going process until the eddies are small enough and molecular viscosity dissipates the turbulent kinetic energy. This behaviour was summarized in a poem by Richardsson in 1922,

Big whorls have little whorls,
which feed on their velocity;
And little whorls have lesser whorls,
And so on to viscosity
(in the molecular sense).

This energy transfer process is called the energy cascade. The energy spectrum of an isotropic and homogenous turbulent flow at high \( Re \), is depicted in Figure 5.1. One may note that the spectrum has a peak at small wave numbers and it is decreasing monotonically for larger wave numbers (i.e. smaller scales). The shape of the energy spectrum is quite general and the behaviour at larger wave numbers (at high \( Re \)) is universal by Kolmogorov’s theory. The range of wave numbers for which the logarithm of the energy spectrum is proportional to the logarithm of the wave-number is the so called inertial sub-range. The proportionality constant is by Kolmogorov’s theory, universal and has the value of \(-5/3\). At even larger wave-numbers dissipation is getting increasingly important, which is manifested in a strong reduction of the energy content with increasing wave-numbers. The wave numbers at which viscous dissipation dominate is called the Kolmogorov scale and is proportional to the kinematic viscosity to the power of \(3/4\). The ratio of the Kolmogorov scale to the integral scale, which characterises the larger eddies, is proportional to the (turbulent) Reynolds number to the power of \(-3/4\). Thus, for high \( Re \)
5.2. COMPUTATIONS OF TURBULENT FLOWS

Figure 5.1. Energy spectra for isotropic turbulence, both axes have logarithmic scales.

due to the range of scales there, has a wide range of turbulent scales. This has a direct impact when solving the system of equations (5.1-5.4) numerically. If no further models are to be introduced all flow scales must be resolved in the solution. This approach is termed as Direct Numerical Simulation (DNS). For a general flow field with the Kolmogorov scale ratio of $Re^{-3/4}$ one finds that the scale ratio for a three-dimensional case is $Re^{-9/4}$. Given also that in order to resolve the time evolution (following the flow with a time step that it takes to convect over the smallest length scale), one would require altogether a resolution better by a factor of $Re^3$ as compared to resolving only the integral length scale.

Obviously, for industrially common Re, DNS is not feasible with today’s computers. A possibility is to solve an averaged set of equations; the so called Reynolds Averaged Navier-Stokes equations (RANS) and model the turbulent part. This approach is rather inexpensive and therefore the most commonly used approach for industrial applications. However, this approach is not intended to and hence cannot capture the dynamics of the flow containing unsteady coherent structures of the order of the size of the integral scale. A possible approach that circumvents this issue is the LES approach. In LES the largest scales of the flow are resolved in the computation while the smallest scales are modelled by a Sub Grid Scale (SGS) model. Since the smallest scales tend to be isotropic for high Reynolds numbers it is easier to find a model that models these small unresolved scales than to find an appropriate model for the stress term in the RANS equation, which represents the turbulent part of the flow field. The approach used in this work is the so called implicit LES, where no explicit sub grid scale model is used. Instead the required dissipative property of the SGS model is provided by the numerical viscosity due to the discretization scheme. This approach will give a solution, which
is an approximation to the Navier-Stokes equation, since as the resolution is improved the contribution of the implicit SGS model reduces to zero. A measure of the resolution can be assessed by studying the local turbulent kinetic energy spectra, which will show to what extent the inertial sub range is resolved and indicate the on-set of the numerical dissipation.

For a more thorough explanation concerning turbulent flows, please refer to a textbook as e.g. Pope S. B. (2000).

5.3. Numerical Methods

The Openfoam code version 1.6 (by OpenCFD Ltd.) was used in this work. Openfoam is an open software, written to solve PDEs based on the finite volume method. The code is open source written in C++ and includes solver routines for simulation of engineering mechanics problems. This enables the user to write their own routines for the specific problem in hand using subroutines and libraries supplied via the code.

For simulation of the flow in the double bent pipe geometry, the compressible flow solver, based on the PISO algorithm described by Issa (1985), was used. For the case with a stationary inlet condition a second-order central difference scheme was used for the spatial discretization, except for the convective term in the momentum equation where a vanLeer Total Variation Diminishing (TVD) limiter, explained in e.g. Hirsch C. (1984), was employed in order to suppress spurious oscillations. For the case with a pulsating inlet velocity the discretization of the convective term was performed with a second-order upwind scheme with a multidimensional cell-limiter. For the temporal discretization the Crank-Nicholson scheme was used.

The number of boundary conditions was given according to the method of characteristics which states how information propagates in a flow domain, i.e. four boundary conditions at the inlet boundary that propagate downstream and a single boundary condition at the outlet that propagates upstream for subsonic flow conditions. In this case the velocity components \((u, v, w)\) and the temperature \((T)\) were set at the inlet and the pressure \((p)\) was specified at the outlet. For the case with a pulsating inlet velocity the outlet boundary was specified as a non-reflective boundary condition to prevent reflections.

5.4. Reduction via a Length Scale Analysis

An option to develop a low-dimensional model for internal flows is to reduce the Navier-Stokes equations (conservation of momentum) into a e.g. 1D- or 2D-model through a length scale analysis, which will be demonstrated for LES data of a double bent pipe. The model reduction is based on neglecting the non-dominating terms of the Navier-Stokes equations projected along the flow path for a specific flow case, where the terms are quantified numerically. The flow path is here defined as the geometric centreline of the geometry or the
central streamline, i.e. the streamline that starts from the centre of the inflow boundary. To show the concept of the method, the flow field can be computed via LES or RANS. The 3D flow field that the reduction is based upon enables an a priori and an a posteriori error estimation of the error the reduced model will give for the specific case and computational method used.

To exemplify the method of the length scale analysis, the following example is studied analytically, which consists of convective and diffusive type terms,

\[
\frac{\partial u}{\partial S} + \frac{\partial u}{\partial \eta} - \frac{\partial^2 u}{\partial S^2} - \frac{\partial^2 u}{\partial \eta^2} = \ldots 
\]  

(5.7)

where the terms are written in curvilinear coordinates, \(S\) in the axial and \(\eta\) in the span wise direction, due to that the flow path for a pipe system or a radial turbine is curved. To compare the terms’ influence they have to be scaled appropriately. For pipe flows the length scales are chosen as the length of the pipe for the axial coordinate and the diameter of the pipe for the span wise coordinate, denoted \(L\) and \(l\) where \(L >> l\), respectively. One may simplify the flow if \(L >> l\), by neglecting the smallest terms. Further reduction is possible if one also excludes other terms. For example, one may neglect all terms except the largest one. In fact, these approximations have been used in fluid mechanics in the past. If one neglects only the smallest terms (often the stream-wise viscous terms in the momentum equations), one may end-up with the so called parabolized Navier-Stokes equations, see e.g. Tannehill et al. (1997). The other extreme case is to only keep the largest term as in the derivation of the classical boundary layer equations. Thus, one introduces the scaling of the coordinates according to the variation of the flow length scales in each direction:

\[
S' = S/L 
\]

(5.8)

\[
\eta' = \eta/l 
\]

(5.9)

The velocity can be scaled with e.g. the velocity of the bulk \((u_b)\). Introducing the scaling into Equation 5.7 gives the following,

\[
\frac{u_b}{L} \frac{\partial u'}{\partial S'} + \frac{u_b}{l} \frac{\partial u'}{\partial \eta'} - \frac{u_b}{L^2} \frac{\partial^2 u'}{\partial (S')^2} - \frac{u_b}{l^2} \frac{\partial^2 u'}{\partial (\eta')^2} = \ldots 
\]

(5.10)

which allows the terms to be quantified from their respective coefficients. The system of equations can now be reduced by e.g. dropping the smallest term, i.e. \(\frac{\partial^2 u}{\partial \eta^2}\) or keeping the largest term, \(\frac{\partial u}{\partial \eta}\). The latter type of scaling was adopted by Prandtl in 1904 to the incompressible Navier-Stokes equations to derive the boundary-layer equations with the assumption that the flow in the boundary-layer is thin in the wall normal direction. The reduced set of equations, i.e.
equations that fall in between the incompressible Navier-Stokes equations and
the boundary-layer equations belongs to a class called the parabolized Navier-
Stokes equations. They all omit the viscous term in the stream wise direction
of the flow. The most important feature is that the parabolized Navier-Stokes
equations are easier to solve and needs less computer time and storage than the
full set of equations, since the solution only depends on upstream conditions.
This approximation is not valid when the basic scale assumption is violated,
for example in the presence of separation bubbles. For compressible subsonic
flow cases where information can travel upstream via pressure waves, the set
of equations have to be solved iteratively to obtain a converged solution on the
whole flow field, which is more costly.

As mentioned above, the flow path for a complex geometry may also have
a complex shape. The terms in the Navier-Stokes equations written in a
Cartesian coordinate system can therefore not directly be quantified, since the
main direction of the flow changes along the flow path. A possibility that is
used here is to transform the equations into a coordinate system where the axial
direction is taken to be one of the coordinate directions. The other coordinates
can be chosen to be orthogonal to the local stream wise direction and also to
each other. The coordinate system chosen for the coordinate transformation
is the orthogonal curvilinear coordinate system where the coordinate lines are
generally curved, as e.g. the streamline in a bent pipe. The transformation is
then performed from the physical domain with the Cartesian coordinate system
to the computational domain (which constitutes a Cartesian system in terms of
the transformed variables) through the curvilinear coordinate transformation,
as shown in Figure 5.2. This approach allows one to quantify the terms in the
Navier-Stokes equations and give an error estimation due to the approximation
that is done while neglecting the different terms (i.e. getting a 1D or a 2D
model).

The transformation onto a curvilinear coordinate system is in general
defined as,

\[ S = S(x, y, z) \]
\[ \eta = \eta(x, y, z) \]
\[ \xi = \xi(x, y, z) \]  \hspace{1cm} (5.11)

where \( S \) is in the direction of the flow path, i.e. the axial direction, \( \eta \) is
chosen to be in the radial direction and \( \xi \) in the azimuthal direction. The axial
component is defined as the distance from the inlet along the flow path. The
radial component is computed as the distance perpendicular to the flow path
and the azimuthal component is computed starting with a chosen direction that
change according to the path line. In practice the direction of the azimuthal
component is computed as the direction normal to the other two coordinate
lines. However, to obtain an one-to-one correspondence between the physical
and the computational domain, the directions of $S$, $\eta$ and $\xi$ must be chosen so that the Jacobian determinant is non-singular. The Jacobian is defined by:

$$J = \frac{\partial (x, y, z)}{\partial (S, \eta, \xi)} = \begin{vmatrix} \frac{\partial x}{\partial S} & \frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \xi} \\ \frac{\partial y}{\partial S} & \frac{\partial y}{\partial \eta} & \frac{\partial y}{\partial \xi} \\ \frac{\partial z}{\partial S} & \frac{\partial z}{\partial \eta} & \frac{\partial z}{\partial \xi} \end{vmatrix} \neq 0 \quad (5.12)$$

The transformation is given by scaling factors ($h_i$), which gives the stretching of the field on the physical domain to be transformed onto the computational domain. If we let a point, $P$ in the physical domain be given by the position vector $\mathbf{r} = xe_x + ye_y + ze_z$, where $e_i$ denotes the basis vectors in each coordinate direction, respectively. Then the coordinate transformation gives $\mathbf{r} = \mathbf{r}(S, \eta, \xi)$. The vectors $\frac{\partial \mathbf{r}}{\partial S}, \frac{\partial \mathbf{r}}{\partial \eta}, \frac{\partial \mathbf{r}}{\partial \xi}$ are tangential to the coordinate curves, i.e. in the direction of the coordinates. Denoting the basis vectors in the computational domain as $e_S, e_\eta, e_\xi$ gives,

$$\frac{\partial \mathbf{r}}{\partial S} = h_S e_S, \quad \frac{\partial \mathbf{r}}{\partial \eta} = h_\eta e_\eta, \quad \frac{\partial \mathbf{r}}{\partial \xi} = h_\xi e_\xi \quad (5.13)$$

with the scaling factors defined as,

$$h_S = \left| \frac{\partial \mathbf{r}}{\partial S} \right|, \quad h_\eta = \left| \frac{\partial \mathbf{r}}{\partial \eta} \right|, \quad h_\xi = \left| \frac{\partial \mathbf{r}}{\partial \xi} \right| \quad (5.14)$$

The scaling factors can directly be computed on the computational domain after the position of each point on the computational domain has been determined on the physical domain with a Cartesian coordinate system.
5. METHODS FOR REDUCED MODELS BASED ON LES

For simplicity we consider incompressible Newtonian fluids. The incompressible Navier-Stokes equations written in vector notation, due to simplicity when deriving the terms in curvilinear coordinates become,

\[
\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} \tag{5.15}
\]

The terms in the incompressible Navier-Stokes equations can now be rewritten into curvilinear coordinates to hold on the computational domain. The general expression for the Navier-Stokes equations on a curvilinear coordinate system can be found in e.g. Tannehill et al. (1997), but are here rewritten to show in which direction the terms act. To obtain the convective terms in curvilinear coordinates explicitly, the convective term, which represents the transport of a property with the flow, can be rewritten as,

\[
(\mathbf{u} \cdot \nabla) \mathbf{u} = \frac{1}{2} \nabla (\mathbf{u} \cdot \mathbf{u}) - \mathbf{u} \times (\nabla \times \mathbf{u}) \tag{5.16}
\]

where the expressions for the curl of a vector and the gradient of a scalar can be found in many textbooks on vector analysis. Expanding the terms gives the following expression for the convective terms in curvilinear coordinates,

\[
(\mathbf{u} \cdot \nabla) \mathbf{u} = \left( u_S \frac{\partial u_S}{\partial S} + u_\eta \frac{\partial u_S}{\partial \eta} + u_\xi \frac{\partial u_S}{\partial \xi} \right) \mathbf{e}_S + \left( u_S \frac{\partial u_\eta}{\partial S} + u_\eta \frac{\partial u_\eta}{\partial \eta} + u_\xi \frac{\partial u_\eta}{\partial \xi} \right) \mathbf{e}_\eta + \left( u_S \frac{\partial u_\xi}{\partial S} + u_\eta \frac{\partial u_\xi}{\partial \eta} + u_\xi \frac{\partial u_\xi}{\partial \xi} \right) \mathbf{e}_\xi \tag{5.17}
\]

i.e. the convection of \( u_S, u_\eta, u_\xi \) on the computational domain. The velocities, \( u_S, u_\eta, u_\xi \) are the velocity components in the axial, radial and azimuthal direction and are computed as,

\[
u_i = \frac{1}{h_i} \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_i}{\partial x_j} v + \frac{\partial u_i}{\partial x_k} w \right), \quad i = 1, 2, 3 \tag{5.18}
\]

The solution to each of the momentum equations will give \( u_S, u_\eta, u_\xi \), respectively. As a first approach we are mainly interested to compute the flow in the axial direction and will therefore mainly focus on the momentum equation in \( \mathbf{e}_S \). The first three terms in the momentum equation denotes convection of \( u_S \) in the three directions corresponding to \( \mathbf{u} \cdot \nabla \mathbf{u} \) in the Cartesian coordinate system. The four remaining terms account for the fact that the coordinate system is curved, which gives a contribution to the convective terms of the momentum equation in \( \mathbf{e}_S \). The fourth and fifth term corresponds to convection of \( u_\eta \) and
5.4. REDUCTION VIA A LENGTH SCALE ANALYSIS

$u_\xi$ in $e_\xi$. The sixth and seventh term correspond to convection of $u_\eta$ and $u_\xi$ in $e_\eta$ and $e_\xi$, which gives a contribution to the momentum equation in $e_\eta$.

The viscous term, which is a diffusion term representing forces due to shear stresses within the fluid, is rewritten as,

$$\nu \nabla^2 \mathbf{u} = \nu \nabla \cdot \Pi_{ij}$$ (5.19)

where the terms of the stress tensor $\Pi_{ij}$ are obtained from computing $\nabla \mathbf{u}$ for the curvilinear coordinates taking into account that the unit vectors are functions of the coordinates, i.e.

$$\Pi_{SS} = \frac{1}{h_S} \frac{\partial u_S}{\partial S} + \frac{u_\eta}{h_S h_\eta} \frac{\partial h_S}{\partial \eta} + \frac{u_\xi}{h_S h_\xi} \frac{\partial h_S}{\partial \xi}$$ (5.20)

$$\Pi_{\eta\eta} = \frac{1}{h_\eta} \frac{\partial u_\eta}{\partial \eta} + \frac{u_S}{h_\eta h_S} \frac{\partial h_\eta}{\partial S} + \frac{u_\xi}{h_\eta h_\xi} \frac{\partial h_\eta}{\partial \xi}$$ (5.21)

$$\Pi_{\xi\xi} = \frac{1}{h_\xi} \frac{\partial u_\xi}{\partial \xi} + \frac{u_S}{h_\xi h_S} \frac{\partial h_\xi}{\partial S} + \frac{u_\eta}{h_\xi h_\eta} \frac{\partial h_\xi}{\partial \eta}$$ (5.22)

$$\Pi_{S\eta} = \frac{1}{h_S} \frac{\partial u_\eta}{\partial \eta} - \frac{u_\eta}{h_S h_\eta} \frac{\partial h_\eta}{\partial S}$$ (5.23)

$$\Pi_{S\xi} = \frac{1}{h_S} \frac{\partial u_\xi}{\partial \xi} - \frac{u_\xi}{h_S h_\xi} \frac{\partial h_\xi}{\partial S}$$ (5.24)

$$\Pi_{\eta S} = \frac{1}{h_\eta} \frac{\partial u_S}{\partial S} - \frac{u_S}{h_\eta h_S} \frac{\partial h_S}{\partial \eta}$$ (5.25)

$$\Pi_{\eta\xi} = \frac{1}{h_\eta} \frac{\partial u_\xi}{\partial \xi} - \frac{u_\xi}{h_\eta h_\xi} \frac{\partial h_\xi}{\partial \eta}$$ (5.26)

$$\Pi_{\xi S} = \frac{1}{h_\xi} \frac{\partial u_S}{\partial S} - \frac{u_S}{h_\xi h_\xi} \frac{\partial h_S}{\partial \xi}$$ (5.27)

$$\Pi_{\xi\eta} = \frac{1}{h_\xi} \frac{\partial u_\eta}{\partial \eta} - \frac{u_\eta}{h_\xi h_\xi} \frac{\partial h_\xi}{\partial \xi}$$ (5.28)

The viscous terms of the momentum equation become,
\[ \nabla \cdot \Pi_{ij} = \frac{1}{\nu} \left[ \frac{\partial}{\partial \eta} (h_\eta h_\xi \Pi_{SS}) + \frac{\partial}{\partial \eta} (h_\eta h_\xi \Pi_{S\eta}) + \frac{\partial}{\partial \xi} (h_\eta h_\xi \Pi_{\xi\xi}) + \frac{\partial}{\partial \xi} (h_\eta h_\xi \Pi_{S\xi}) - \frac{h_\eta h_\xi \Pi_{\xi\xi}}{\nu} \right] e_\xi \\
+ \frac{\partial}{\partial \eta} (h_\eta h_\xi \Pi_{\eta S}) + \frac{\partial}{\partial \eta} (h_\eta h_\xi \Pi_{\eta\eta}) + \frac{\partial}{\partial \xi} (h_\eta h_\xi \Pi_{\xi\eta}) + \frac{\partial}{\partial \xi} (h_\eta h_\xi \Pi_{\xi\xi}) - \frac{h_\eta h_\xi \Pi_{\xi\xi}}{\nu} \right] e_\eta \\
+ \frac{\partial}{\partial \eta} (h_\eta h_\xi \Pi_{\xi S}) + \frac{\partial}{\partial \eta} (h_\eta h_\xi \Pi_{\xi\xi}) + \frac{\partial}{\partial \xi} (h_\eta h_\xi \Pi_{\xi\eta}) + \frac{\partial}{\partial \xi} (h_\eta h_\xi \Pi_{\xi\xi}) - \frac{h_\eta h_\xi \Pi_{\xi\xi}}{\nu} \right] e_\xi \] (5.29)

The first three of the viscous terms in the momentum equation for \( e_\xi \), corresponds to \( \nabla^2 u \) in the Cartesian coordinate system. The remaining terms compensate for the fact that the coordinate system is curved in a similar way as for the convective terms, i.e. the fourth and fifth term correspond to shear stresses of \( u_\eta \) and \( u_\xi \) in \( e_\xi \), the sixth and seventh term correspond to normal stresses in \( e_\eta \) and \( e_\xi \) that influence the viscous term in \( e_\xi \).

Quantifying the different convective and viscous terms will show the error a reduction of each term will give. However, even though the viscous terms are most certainly smaller than the convective terms, not all of them can be neglected if one would like to account for viscous effects in the span wise plane. The pressure terms, i.e. the gradient of the pressure has to balance the inertia terms and has therefore to be included in the reduced model.

5.5. Reduction through Galerkin Projection onto POD Modes

A method is presented for reduction of the Navier-Stokes equations via Galerkin projection onto Proper Orthogonal Decomposition (POD) modes. The POD modes constitute an orthonormal set of modes that is generated on the basis of a set of instantaneous flow field data. The POD base has the property that it is optimal in the sense that it requires the smallest number of terms to express the kinetic energy of the flow field. The idea with the Galerkin projection is to reduce the original system of PDEs to a set of Ordinary Differential Equations (ODE). The modelling approach allows continuing the time marching of detailed LES in the parameter space of the problem (i.e. variations in flow parameters and/or boundary conditions). As input to the model one needs a set of LES data, which constitutes the basis for deriving the POD modes. In the following we discuss the generation of the POD modes in more detail.

The Galerkin projection converts a partial differential equation into a set of ODEs by projecting the functions defining the original equations onto a subspace. The POD modes will in this example represent the subspace for the projection. However, any other basis function can be used for the purpose. For
5.5. REDUCTION THROUGH GALERKIN PROJECTION ONTO POD MODES

Illustration of the technique, the POD is computed for a double bent pipe, a radial turbine and a ducted orifice plate. The Galerkin projection is performed for the double bent pipe only. The approach is thoroughly described by Smith et al. (2005) but will also be described here for consistency.

The flows we are dealing with in internal flow components for engine applications are turbulent by nature. Turbulent flows contain large scale structures denoted as coherent structures. Coherent structures are explained by Holmes et al. (1996) as repeatedly appearing forms that are spatially coherent, temporally evolving, energetically dominant vortical motions of the flow. POD was introduced by Lumley (1967) in the context of turbulent flows with the aim to detect these coherent structures. POD is a decomposition method that decomposes the flow field into one spatial part and one temporal part, Equation 5.31. The spatial part consists of so called POD modes, which describe the flow structures. The temporal part consists of the so called POD or time coefficients, which describe the time evolution of the flow structures. POD decomposes the flow field in an optimal manner with respect to energy, i.e. the energy content captured by a limited number of modes is larger than if the flow field was projected onto any other flow structures (modes) formulated as,

\[
\max_{\varphi \in L^2(0,1)} \frac{\langle |(u(x,t), \varphi(x))|^2 \rangle}{\| \varphi(x) \|^2} \tag{5.30}
\]

where the inner product \( \langle f, g \rangle = \int_0^1 f(x)g^*(x)dx \) and \( \| f \| = (f, f)^{1/2} \), i.e. the \( L^2 \)-norm. The first POD mode is nothing else but the structure that represents the mean flow field. The consequent modes (ordered in their relative energy content) have a smaller and smaller contribution to the total energy of the flow. In flows with coherent structures, the lower numbered modes contain these coherent structures.

Reconstructing the flow field with a limited number of POD modes allows one to study coherent structures independent of the turbulent part of the flow field. For a statistically stationary flow field and when the coherent structures have a higher energy content than the turbulent kinetic energy, the POD will decompose the flow into a part related to the mean flow, a part with coherent structures that evolve in time and a turbulent (incoherent) part as,

\[
u(x, t_k) = \underbrace{a_0 \varphi_0(x)}_{\text{mean}} + \sum_{j=1}^J a_j(t_k) \varphi_j(x) + \sum_{j=J+1}^\infty a_j(t_k) \varphi_j(x) \tag{5.31}
\]

\[\text{coherent} \quad \text{incoherent} \quad \text{fluctuations}\]
where \( j \) denotes mode number, \( k \) the time step, \( a(t) \) the time coefficients, which are usually called the chrono modes and \( \varphi(x) \) the POD modes or in other words the topo modes.

Historically POD has mainly been used by experimentalists to study large scale energetic features of turbulent flows measured with e.g. hot-wire probes, Bakewell & Lumley (1967) and more recently using Particle Image Velocimetry (PIV), Kostas et al. (2005). Kostas et al. (2005) showed how POD can be applied to other physical quantities than the velocity, as vorticity in their case to better describe the dominant vortical structures of the flow. However, POD has also been applied to data sets obtained from numerical computations as shown by Moin & Moser (1989) and Manhart & Wengle (1993). Moin & Moser (1989) applied POD onto a data set from a DNS of a turbulent channel flow to characterize the flow. Due to the computer expense of DNS computations Manhart & Wengle (1993) performed POD analysis onto data from LES computations on the flow over sharp-edged obstacles as a cube at a high Reynolds number (\( \approx 50000 \)) with the aim to study the flow for geometries of engineering interest.

In addition to the POD approach there are several other alternative ways of decompositions, e.g. Fourier modes or Koopman modes described by Rowley et al. (2009), which decouples the frequency components of the flow more effectively than the POD modes. Cazemier et al. (1998) compared the energy content for the POD and Fourier modes which clearly shows that the first POD modes contain significantly more energy than the Fourier modes. A comparison of the POD and the Koopman modes was performed by Muld (2010), who showed their behaviour for flows over a cube.

Cazemier et al. (1998) performed an extensive study of the performance of a low-dimensional model constructed via Galerkin projection of the incompressible Navier-Stokes equations onto POD modes. The low-dimensional model was constructed from 20, 40 and 80 modes obtained from 700 snapshots of the velocity field for a dataset from a DNS simulation for driven cavity flows. Cazemier et al. (1998) showed that the model with the higher number of modes converges to a statistical equilibrium around a state with a too high energy content compared to the DNS results, since the conservation of energy is not fulfilled due to the truncation error within the reconstruction. A dissipative closure model was therefore added. Rowley et al. (2004) showed that a reduced model for flows with strong coherent structures can be derived by the Galerkin projection based on only the POD modes that represent these coherent structures. The approach was shown for a flow in a cavity represented by four POD modes. Recently, Navarro et al. (2010) used a reduced model with the leading eigen-vectors, the eigen-values with the largest real parts, as the basis for the Galerkin projection to study the Rayleigh-Bénard convection, i.e. convection occurring in a plane of fluid heated from below, in a cylindrical geometry with a flow generated by two counter rotating disks. The reduced
model based on 41 eigenvectors was shown to capture complex dynamics of the flow with an error of less than 0.1.

5.5. Computing the POD Modes

To compute the POD modes, the maximization problem, Equation 5.30, can be recast into an eigen-value problem as e.g. shown in Holmes et al. (1996), whose kernel is the averaged autocorrelation function. For the computation, a series of statistically independent time steps, snapshots, of a discrete flow field have to be saved. The flow fields are here written into a matrix as,

$$\mathbf{U}_i = \begin{bmatrix} u_i(x_0, t_0) & \ldots & u_i(x_0, t_M) \\ \vdots & \vdots & \vdots \\ u_i(x_N, t_0) & \ldots & u_i(x_N, t_M) \end{bmatrix}$$

(5.32)

where \(N\) denotes the number of nodes and \(M\) the number of snapshots. The discretized eigen-value problem can be solved with respect to the domain \(\Omega\), Equation 5.33, where the eigen-vectors are the actual flow structures, i.e. the POD modes, or with respect to the time series \(T\), Equation 5.34.

$$R_{\Omega} \varphi_l(x_n) = \lambda_l \varphi_l(x_n)$$

(5.33)

$$R_{T} q_k(t_m) = \mu_k q_k(t_m)$$

(5.34)

The autocorrelation matrices are defined as,

$$R_{\Omega} = \sum_{i=1}^{3} \left( \mathbf{U}_i \mathbf{U}_i^T \right)$$

(5.35)

giving a \(N \times N\) matrix or as,

$$R_{T} = \sum_{i=1}^{3} \left( \mathbf{U}_i^T \mathbf{U}_i \right)$$

(5.36)

giving a \(M \times M\) matrix, respectively. The size of the eigen-value problem to solve, is set according to the way the autocorrelation function is computed. For a LES the usual case is that \(M << N\), which makes it preferable to solve the eigen-value problem with respect to time, defined according to Equation 5.34. Since the kernel of the eigen-value problem is positive and symmetric the eigen-vectors are real, positive and orthogonal, \((\varphi_j, \varphi_k) = \delta_{jk}\). For this case, the POD modes have to be computed from the eigen-vectors \(q_k(t_m)\) and the velocity fields according to,
\[ \varphi_k(x_n) = \sum_{m=0}^{M} q_k(t_m) u_i(x_n, t_m) \]  
\[ (5.37) \]

However, the orthogonality of the vectors will not persist and therefore the Gram-Schmidt Orthogonalisation method was applied to the newly computed POD modes. The Gram-Schmidt Orthogonalisation method finds orthonormal vectors that span the same subspace as the original vectors. The time coefficients for the POD modes are after the orthonormalisation obtained as,

\[ \alpha_k(t_m) = (\varphi_k(x_n), u_i(x_n, t_m)) \]  
\[ (5.38) \]

The square of the time averaged coefficients, correspond to the eigen-values of the eigen-value problem, hence the eigen-values represent the amount of energy of \( u_i(x_n, t_m) \) projected onto \( \varphi_k(x_n) \). The POD modes can then be ordered in descending order according to their energy content, \( \lambda_k > \lambda_{k-1} > ... \).

5.5.2. The Galerkin Projection

The aim of the Galerkin projection is to convert a PDE into a set of ODEs by projecting the functions defining the original equations onto a subspace, in this case the POD modes. To simplify the problem somewhat, we will use the incompressible Navier-Stokes equations including the continuity equation, Equation 5.40 and 5.39, respectively. However, as shown by Rowley et al. (2004) a similar approach can be used to reduce the compressible Navier-Stokes equations.

\[ \frac{\partial u_i}{\partial x_i} = 0 \]  
\[ (5.39) \]

\[ \frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} + \frac{1}{\rho} \frac{\partial p}{\partial x_i} - \nu \frac{\partial^2 u_i}{\partial x_j^2} = 0 \]  
\[ \text{NS} \]  
\[ (5.40) \]

Once the POD modes are determined one may express the physical quantities in terms of the POD expansion, Equation 5.31. Keeping only the most energetic terms, which represents a considerable portion of the total kinetic energy (say some 95 %). By inserting the expansion into the Navier-Stokes equations, one obtains a set of equations for the POD coefficients, which are only time dependent. Forming the scalar product of the Navier-Stokes equations with the POD modes and utilizing the fact that the modes are orthogonal to each other, the problem reduces into a system of ODEs for the POD coefficients,

\[ \frac{d\alpha_k}{dt} + (NS(x), \varphi_k(x)) = 0 \]  
\[ (5.41) \]
where the transient term is obtained through,

\[
\left(\frac{\partial}{\partial t}\sum_{j=0}^{M} \alpha_j \varphi_j(x), \varphi_k(x)\right) = \{(\varphi_j, \varphi_k) = \delta_{jk}\} = \frac{d\alpha_k}{dt} \tag{5.42}
\]

Thus, the coefficients can be integrated in time and the solution can thereby be reconstructed as a function of time. Hence, instead of solving a system of hyperbolic PDEs, one can simply integrate a few ODEs in time and save substantial computational effort. The number ODEs that have to be solved is dependent on the number of modes needed to obtain a predictive low-dimensional model, which is case dependent. When the POD modes have been determined for a certain flow condition and inserted into the model, it can be used to compute the flow at nearby conditions. The accuracy of the model will then depend on the number of modes used for the decomposition and the deviation from the original flow condition. The error introduced by the approximation of the infinite-dimensional evolution equation, Equation 5.40 can be determined by the residual of the decomposition, i.e. \(\sum_{j=N+1}^{\infty} \alpha_j \varphi_j\). However, as pointed out in Smith et al. (2005) the best model is not necessarily obtained by keeping the most energetic modes. Since the most energetic modes might not be the most essential ones to describe the dynamics of the flow.

The time averaged part of the Navier-Stokes equations is known from the LES and does not need to be modelled by the Galerkin projection, thus only the fluctuating part has to be modelled. This will also give the advantage that the convective term can be divided into a linear and a non-linear part. The linear part will later be used for stabilising the system of ODEs. The fluctuating part of the Navier-Stokes equations can be derived by subtracting the averaged equations, i.e. the RANS equations, Equation 5.44, from the instantaneous Navier-Stokes equations. Rewriting the momentum equation after decomposing the physical quantities into a mean and a fluctuating part, i.e. \(u = \bar{u} + u'\) and \(p = \bar{p} + p'\) gives the following equations,

\[
\frac{\partial (\bar{u} + u')}{\partial t} + \frac{\partial (\bar{p} + \bar{u}'u')}{\partial x_j} + \frac{\partial (\bar{u}'u_j)}{\partial x_j} + \frac{\partial (u'u'_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}}{\partial x_i^2} \tag{5.43}
\]

Averaging gives the RANS equations,

\[
\frac{\partial \bar{u}}{\partial t} + \frac{\partial (\bar{u}\bar{u})}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}}{\partial x_i^2} - \frac{\partial (\bar{u}'u'_j)}{\partial x_j} \tag{5.44}
\]

which subtracted from the instantaneous equations, Equation 5.43, results in,
decomposition of the pressure by taking the inner product of the Poisson equation with each mode from the \( q \) for the divergence of the momentum equation, Equation 5.40, and using the continuity equation via the PISO loop as suggested by Issa (1985). By taking the same principle as when solving the Navier-Stokes equations within a CFD computation via the PISO loop as suggested by Issa (1985). By taking the divergence of the momentum equation, Equation 5.40, and using the continuity condition, Equation 5.39, the following equation is obtained for the pressure with respect to the velocity components, i.e. the Poisson equation,

\[
\frac{\partial^2 \rho'}{\partial x_i^2} = -2 \rho \frac{\partial^2 (\bar{\pi}_j u'_i)}{\partial x_i \partial x_j} + \frac{\partial^2 (u'_i u'_j)}{\partial x_i \partial x_j} \tag{5.47}
\]

The Poisson equation can be expanded in a similar manner as the Navier-Stokes equations. However, to be able to compute the coefficients \( \beta_q \), i.e. for the \( q \) modes used for construction of the low-dimensional model for the decomposition of the pressure, \( q \) equations are needed. These can be obtained by taking the inner product of the Poisson equation with each mode from the decomposition of the pressure \( \psi_c \) according to,

\[
\sum_{q=1}^{M} \beta_q \left( \frac{\partial^2 \psi_c}{\partial x_i \partial x_i}, \psi_c \right) = -2 \rho \sum_{l=1}^{M} \alpha_l^j \left( \frac{\partial^2 \bar{\pi}_j \psi_c}{\partial x_i \partial x_j}, \psi_c \right) - \sum_{l=1}^{M} \alpha_l^j \left( \frac{\partial^2 (\bar{\pi}_j \psi_c)}{\partial x_i \partial x_j}, \psi_c \right) + \left( \frac{\partial^2 (u'_i u'_j)}{\partial x_i \partial x_j}, \psi_c \right) \tag{5.48}
\]

which gives the following system of linear equations to solve,
5.5. REDUCTION THROUGH GALERKIN PROJECTION ONTO POD MODES

\[ B\beta = a \]  \hspace{1cm} (5.49)

The system of linear equations is solved with a LU-decomposition of the matrix \( B \) and through backward and forward substitution. This gives the \( \beta \)-coefficients, which are used to represent the pressure term when integrating the ODEs in time. The 3D velocity field can then be reconstructed in time.

5.5.3. Solving the System of ODEs

The Galerkin projection gives a coupled time dependent non-linear system of ODEs. The system of ODEs is solved by the explicit fourth-order Runge-Kutta method, chosen for its ease of implementation while still giving an accurate result for relatively large time steps. However, since a system of ODEs is inherently unstable, there is a need for damping to get a stable solution. Damping was added to the linear terms, which were found to be the main cause of the instability of the system.

The system of ODEs can be written with the linear and non-linear terms apart as,

\[
\frac{dx}{dt} = Ax + Gx^2 + C \]  \hspace{1cm} (5.50)

where the behaviour of the linear part of the system of ODEs is determined by the eigen-values of \( A \), \( (\lambda_i) \), i.e.

- If any \( re(\lambda_i) > 0 \) \( \Rightarrow \) the system is unstable
- \( im(\lambda_i) \neq 0 \) \( \Rightarrow \) oscillatory behaviour

The damping was added to the linear part of the system of ODEs by setting its eigen-values with positive real parts to zero.
CHAPTER 6

1D Engine Simulation Results

Within the first part of the thesis, the turbine on-engine behaviour is presented, including both numerical and experimental results. Numerical simulations were conducted for the entire engine system both in stationary and transient operation and validated with the experimental results.

6.1. Instantaneous On-Engine Turbine Efficiency

The scope of this study is to calculate the instantaneous on-engine turbine efficiency from measured and simulated engine data. Since instantaneous mass flow rate and temperatures across the turbine are difficult to measure, this information is taken from simulation results from a calibrated engine model. The calculations are performed as a step towards generating turbine performance maps from engine measurements.

The turbine efficiency was determined for two cases with an engine torque of 1850 Nm at 1200 and 1400 rpm. The results are only shown for the 1400 rpm case since both cases show similar on-engine turbine behaviour.

6.1.1. Model Validation

To verify the simulation results, simulation data from the engine model was validated against experimental data from the fast and slow measurement system. To get the model to quantitatively reflect the measured engine performance, the model had to be calibrated with respect to turbine efficiency. This is due to the fact that the turbocharger speed was too low, resulting in an under predicted turbine output power, driving the compressor, which resulted in a too low boost pressure, thus less energy to the turbine. The turbine efficiency multiplier, which increases the efficiency of the turbine, hence its power output, was increased by 10 % at high engine speed, i.e. at 2000 rpm and 50 % at low engine speed, i.e at 1000 rpm. Figure 6.1 shows a comparison between simulated and measured crank-angle resolved phase-averaged data from 100 consecutive cycles for the pressure traces before the turbine and turbo shaft acceleration, respectively. The pressure trace shows the blow-down pulses from cylinder 1-3 and three pulses from cylinder 4-6 due to cross flows at the turbine rotor. Figure 6.2 shows the temperature and mass flow traces from simulations in the exhaust manifold branch for cylinder 1-3 upstream of the turbine.
6.1. INSTANTANEOUS ON-ENGINE TURBINE EFFICIENCY

6.1.2. Results from Turbine Efficiency Calculations

The instantaneous on-engine turbine efficiency is calculated directly according to Equation 2.11, i.e. as the ratio between actual work output, extracted power, and the work from isentropic expansion of the gas over the turbine, Figure 6.3. Comparison of the efficiencies for different blow down phases, Figure 6.4, shows that the turbine efficiency trace for the blow down phases for cylinder 1, 2 and 3 has a similar behaviour as for cylinder 4, 5 and 6, which are each others counterparts of the exhaust manifold branches. The
6.1D ENGINE SIMULATION RESULTS

Figure 6.3. Crank-angle resolved turbine efficiency and extracted and isentropic power output. The pulses are numbered according to each cylinder’s blow down pulse.

average, output power weighted, efficiencies for the different cylinders blow down phases for each manifold branch and for each manifold branch evaluated from stationary turbine maps are presented in Table 2. The results show that the average efficiencies are higher for exhaust pulses from cylinder 4, 5 and 6, which blow into the outer volute entry, i.e. to the entry furthest away from the compressor, than from cylinder 1, 2 and 3. This holds for each cylinder pair with symmetrical exhaust manifold branches, i.e. cylinder 1 and 6 where cylinder 6 gives a higher efficiency than cylinder 1, 2 and 5 and cylinder 3 and 4 where cylinders 5 and 4 give a higher efficiency than cylinder 2 and 3, respectively. Worth noting is that the outer cylinders, cylinder 6 and 1 give a higher efficiency than cylinder 2 and 5. The same holds for cylinder 2 and 5 compared to cylinder 3 and 4. Comparing the isentropic power over the turbine for each cylinder shows that the isentropic power differs at most +/- 2 % between the different cylinders, where cylinder number 3 gives the highest and cylinder number 5 the lowest isentropic power over the turbine. The power from the turbine differs with as much as +5 % for cylinder number 6 to -6 % for cylinder number 3.

6.1.3. Discussion

The turbine efficiency had to be manipulated to get the performance of the virtual engine to reflect the measured engine performance. However, there are a number of uncertainties that can be modelled incorrectly and then corrected by manipulating the turbine performance in the model. Primarily it is uncertain
6.1. INSTANTANEOUS ON-ENGINE TURBINE EFFICIENCY

Figure 6.4. Efficiency comparison for the two cylinder banks. The derived efficiency is adjusted 360 crank-angle degrees for the blow down pulses from cylinder 5, 3 and 6. Cylinder numbering at the top of the figure.

Table 2. Average, output power weighted, turbine efficiencies for the six different cylinder's blow down phases, for each manifold branch and for each manifold branch evaluated from stationary turbine maps.

<table>
<thead>
<tr>
<th>Cyl</th>
<th>$\eta_{cyl}$</th>
<th>$\eta_{branch}$</th>
<th>$\eta_{stat.map}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.75</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.73</td>
<td>0.72</td>
<td>0.79</td>
</tr>
<tr>
<td>3</td>
<td>0.69</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.74</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>0.77</td>
<td>0.76</td>
<td>0.79</td>
</tr>
<tr>
<td>6</td>
<td>0.78</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

if the energy into the turbine is correctly modelled; since it was not possible to measure the instantaneous temperature and mass flow rate upstream the turbine. Secondly, the turbine performance data is measured under steady flow conditions with no secondary flow structures, which is not the case when employed on an IC-engine.

The method to compute the on-engine instantaneous efficiency seems uncertain with a lot of possible errors due to the modelling approach and the fact that there is a phase shift between the different signals that was not considered.
Figure 6.5. Measured turbo shaft acceleration, pressure ratio between the two turbine volute inlets and simulated mass flow in the orifice connection. Local peaks in the turbo shaft acceleration are marked with arrows.

However, the asymmetric on-engine turbine behaviour is in agreement with results presented by Dale (1990) who measured the efficiency for a twin-entry turbine in a flow bench under different partial admission conditions. The results presented in Table 2, also show a systematic behaviour of the turbine where the exhausts from the outer cylinders with a long exhaust manifold gives a higher turbine efficiency than from the inner cylinders, which are connected to the turbine with a shorter exhaust manifold. An explanation for this could be that exhausts from outer cylinders with long exhaust manifolds will have a more uniform flow than from the inner cylinders with shorter exhaust manifolds, thus giving higher turbine efficiency.

The measured and simulated rotational shaft acceleration of the turbo shows noticeable differences, Figure 6.1. Two peaks at the decreasing phases can be noticed from the measurements, which the simulation results do not show. These peaks can be seen for different cycles and for each cylinder’s blow down phase, which implies that it is a repetitive behaviour of the turbine. A
6.2. MODEL-BASED TURBINE MAP

The scope of this study is to compare simulation results using an extrapolated turbine map based on measurements and a map generated via the software Rital based on semi-empirical models and calibrated to the turbine measurements provided by the turbo manufacturer. The measured map is extrapolated according to a curve fit based on normalized efficiency and mass flow rate as a function of normalized blade speed ratio, Figure 4.2. The comparison is made within an engine simulation software, simulating a load transient to use the maps in a large operational range. A sketch of the engine modelled is shown in Figure 4.3.

6.2.1. Turbine Modelling with RITAL

The Rital turbine model was calibrated against measured manufacturer turbine performance data with respect to efficiency and mass flow rate. Figure 6.6 shows the results for two out of eight speed lines used. Except turbine performance data the throat opening area is a measure which the model was verified with. Simulation data gave a throat opening area of 2231 mm² while the actual opening area is 2261 mm². The model generated with Rital gave negative efficiencies and unreasonable mass flow rates at high speed and low
6.1D ENGINE SIMULATION RESULTS

Figure 6.6. Efficiency and reduced mass flow rate from calibration of the Rital turbine model, dotted lines with a star, compared to measured turbine data, solid lines with a plus as a function of blade speed ratio at two reduced speeds.

pressure ratios. This is due to that the models are only verified and written to model turbine performance under stationary conditions. High speeds and low pressure ratios can only be achieved when suddenly less amount of fuel is injected into the engine, i.e. in a negative load transient. These data points were therefore not included in the full resolution turbine maps.

6.2.2. Simulation results compared to measurements

The transient presented is from approximately 20 \% load (400 Nm) to maximum load in shortest possible time at constant speed, 1400 rpm. The signal to the engine’s Electronic Control Unit (ECU) was instantaneously increased to its maximum at start of the load transient, cycle 21. The injected amount of fuel was then controlled by the ECU with respect to boost pressure build up to prevent high amounts of smoke. Figure 6.8 shows the measured brake torque and the simulated and measured cycle-averaged mass flow rate air and pressures and temperatures upstream and downstream the turbine after manipulating the turbine efficiency. Full load, i.e. 1800 Nm, was reached after 28 cycles, which corresponds to 2.4 s. The speed fluctuations during the measurements were +/- 38 rpm, which corresponds to 2.8 \% of the desired engine speed due to the slow response of the engine brake. The traces of pressure and mass flow rate follow
the measurements fairly well. However, due to the slow temperature sensors used to measure the temperature upstream and downstream the turbine, the simulated and the measurements are not showing the same behaviour during the transient. The turbine efficiency multiplier was tuned to get the turbine power output, which affects the turbocharger speed that can be measured with high accuracy, in the simulations to correspond to the measurements, Figure 6.7. However, the large multiplier needed could be due to other errors that the change in turbine efficiency can overshadow, since it affects the whole gas exchange system.

To be able to study the turbine modelling and especially how well the model predicts the power output from the turbine, the simulated energy input to the turbine must be correct, both cycle-averaged and crank-angle resolved. Due to measurement difficulties of the instantaneous temperature and mass flow rate upstream the turbine, only crank-angle resolved data of the pressure traces were possible to measure. Figure 6.9 shows three cycles of measured and
simulated pressure traces upstream the turbine and the turbocharger speed crank-angle resolved at the start of the transient. The first cycles during a load transient are believed to be the most challenging to predict since these operational conditions are furthest from stationary engine operation.

6.2.3. **Turbine map comparison**

The full resolution maps generated via the semi-empirical modelling tool calibrated with turbine performance from a steady flow rig and the full resolution map that is an extrapolation of the measurements are shown in Figure 6.10 and 6.11, which shows efficiency versus reduced mass flow rate and pressure ratio. On top of the maps, the measured turbine performance data points are marked and the operational line for cycle 23 from the simulations is plotted, i.e. the third cycle after the start of the load transient. The operational line shows that the turbine works in a large range over a single cycle and most of the time outside the measured data points. The Rital map shows a higher maximum efficiency area, 2.5 %-units and the high efficiency area is larger than for the GT-Power map.

The turbine efficiencies from the two maps over the cycle in the start of the load transient are shown in Figure 6.12, which shows simulated turbine efficiency crank-angle resolved, mass flow rate and pressure ratio across the turbine for cycle 23 during the simulated load transient. The model generated turbine map gives larger turbine efficiency variations than the map generated directly via extrapolation of the measured turbine performance with efficiency.
dips down to 20%. These efficiency dips are not affecting the overall turbine performance significantly, due to that the low efficiency regions coincide with the low mass flow rates and low pressure ratios, i.e. at low exhaust energies.

The two full resolution turbine maps were used to model the turbine within the model for transient engine operation. The two set of turbine maps were used directly, without manipulation of the turbine performance parameters as needed earlier to get the simulation results to correspond to measurements. Figure 6.13 and 6.14 shows the simulation results, i.e. the turbine efficiency and turbocharger speed cycle averaged. In the beginning of the transient, the model generated by Rital gives 1 %-unit lower turbine efficiency. At the end of the transient the Rital model gives 1 %-unit higher turbine efficiency than the turbine maps obtained from measured data. This efficiency difference is not enough to affect the turbocharger speed significantly. During the entire run the turbine power output is clearly under predicted, which gives a lower turbocharger speed compared to the measurements, Figure 6.8.

6.2.4. Discussion

The simulation results with the two turbine maps show similar results as shown in Figure 6.14. The cause for this is that the turbine performance data from the manufacturer is measured for a relatively large operational range as Figure 4.2 shows, which was used for calibrating the semi-empirical turbine model.
As shown in Figure 6.8, the calibration of the engine model when run in transient mode needed adjustment of the turbine efficiency, i.e. the output power, to above 30% during the transient. This had to be done even though steady-flow turbine performance data measured with a dynamometer obtained from the turbo manufacturer is considered relatively accurate. According to the
6.2. MODEL-BASED TURBINE MAP

Figure 6.12. Turbine efficiency for the two differently generated turbine maps for cycle 23, i.e. at the start of the transient and the mass flow rate and pressure ratio across the turbine.

In the turbine manufacturer the measured turbine performance data, i.e. efficiency and mass flow rate, is measured with a repeatability of +/- 0.75 respectively +/- 0.5%. The predicted peak pressure before the turbine, Figure 6.9, is lower than the measured peak pressures even though predicted average pressure is similar to the measured pressure. This implies that the predicted energy to the turbine is lower than in reality and a need for a turbine efficiency multiplier greater than one. Another cause is the lower temperature of the exhaust gases going into the turbine, Figure 6.7.

The turbine used for this investigation has a twin-entry volute. Turbine performance measurements are performed at full admission by the turbocharger manufacturer even though a twin-entry turbine on an IC-engine works mostly under partial admission. Differences of a couple of percent in turbine efficiency on the engine have been shown, Section 6.1. However, the need of increasing the turbine power output by 30% can, however not be explained by the fact that the turbine performance data were measured at equal admission.
Figure 6.13. Turbine efficiency cycle-averaged during the transient for the two differently generated turbine maps.

Figure 6.14. Turbine speed cycle-averaged during the transient for the two differently generated turbine maps.
6.3. Simulation of a Two-Stage Turbocharged Engine

As written by Langridge and Fessler (2002), future heavy-duty diesel engines will be equipped with more complex gas exchange systems, which is where we are today with turbo compound, two-stage turbocharging (for passenger car engines), high EGR rates, variable valve timings, etc. As focus is more towards transient engine performance there is a need to optimize the engine for transient engine test cycles through simulations. It is therefore desirable to have predictive models with a high degree of detail as a tool in system design, engine optimization and engine control.

In this work, a 1D model of an engine with a two-stage turbocharger system including a high pressure-loop has therefore been validated with experimental data. The engine configuration is sketched and its installation shown in Figure 4.4 and Figure 4.5, respectively. The main focus has been on transient operation where the transient studied is a load transient at constant engine speed, which is difficult to predict due to the turbocharger response time. The model has also been used to try different VGT positions for a rapid transient response.

6.3.1. Experimental Results

The transient studied is a load transient from approximately 20% load (400 Nm) to maximum load in shortest possible time at a constant speed of 1400 rpm. The ECU from the base engine configured for EURO 3 emission legislation was used, which limits the injected amount of fuel in relation to the available amount of air to prevent smoke. During the measurements, the VGT was adjusted to give 30% EGR before and after the load transient and held open during the transient to get the exhaust gases to expand over the twin-entry turbine at the high pressure stage, which was thought intuitively to give the fastest transient response. The EGR valves were held open before and after the load transient and at different positions during the load transient to show its effect on the response time. The measured brake torque and engine speed are shown in Figure 6.15, which shows that the brake torque reaches 1900 Nm approximately 2 s faster with the EGR valves closed compared to when they are open. The response time until 1200 Nm is similar for different EGR valve positions, since the injected amount of fuel is not limited by the available amount of air, which is due to the response time of the engine brake and not the engine itself. The engine brake was therefore set to start braking just before the power demand from the engine, which is seen as a dip of the engine speed just before the start of the transient.

The two turbochargers in the system will interact with each other depending on the different EGR valve and VGT positions. The turbocharger speed and the effect from different EGR valve positions are shown in Figure 6.16. During the transient, when the VGT is held open, the speed of the HP-turbo raises more rapidly than for the LP-turbo. However, after the transient when
6. 1D ENGINE SIMULATION RESULTS

Figure 6.15. Measured brake torque at different EGR valve positions including engine speeds, left and simulated EGR rate with the open VGT during the transient, right.

Figure 6.16. Measured turbocharger speeds with different EGR valve positions during the transient.

the VGT is closed to get an EGR rate of 30 %, the speed of the HP-turbo decreases abruptly and the speed of the LP-turbo raises again.
6.3. SIMULATION OF A TWO-STAGE TURBOCHARGED ENGINE

6.3.2. Results from Simulations

Stationary engine operation - The model was first calibrated for stationary operation at high load and 30% EGR. To get close agreement to measured data, the turbine performance had to be adjusted with up to 10% with respect to both efficiency and size to give satisfactory results with respect to time averaged pressures throughout the gas exchange system and turbocharger speeds. Figure 6.17 and 6.18 show crank-angle resolved pressures before and after the high pressure turbine, with and without the EGR valves and the VGT open. The simulated pressure peaks before the high pressure turbine are lower than the measured and the pressure trace for the case with open EGR valves compared to closed EGR valves is not as well matched at the crank-angle intervals where cross flows within the twin-entry volute occur. To obtain good agreement with the measured CA-resolved data with respect to pressure amplitudes, especially after the high pressure turbine, 70% of the volume of the volutes for both turbines was added to the turbine models, respectively.

The turbocharger acceleration of the two turbochargers is shown in Figure 6.19. A difference in amplitude and in phasing can be seen. The phase difference for the LP turbo is 8 to 12 CAD.
Figure 6.18. Crank-angle resolved pressures downstream the high pressure turbine, with and without EGR.

Figure 6.19. Crank-angle resolved high pressure and low pressure turbocharger acceleration, with and without EGR, fully open VGT.
6.3. SIMULATION OF A TWO-STAGE TURBOCHARGED ENGINE

Figure 6.20. Compressor performances for 1000-2000 rpm at high load including 30% EGR, indicated with white dots. High pressure compressor, upper figure and low pressure compressor, lower figure.

The performance of the compressors is shown in Figure 6.20 for 1000-2000 rpm of engine speed at high load including 30% EGR. The HP-compressor gives a high efficiency at the entire rpm range. However, in order to not reach too high EGR rates, the VGT had to be held open at high engine speeds, which resulted in poor compressor efficiency. This indicates that the HP-turbine should be larger for high engine speeds or have the possibility to be by-passed to prevent a high engine back pressure.

Transient engine operation - For the transient simulations, the model was first run in steady-state for the low load case to achieve stable starting conditions for the transient, i.e. with respect to temperatures and turbocharger speeds. The engine was then run during a load transient, i.e. at an instant load demand at a constant engine speed. Since the turbocharger model is in focus for the study...
the in-cylinder conditions with respect to heat release rate and injected amount of fuel were set from measurements. For the parts of the transient where the start of injection was changing, the heat release rate was computed and used in the model for each cycle. That is at the start of the transient where the relation between the pre-mixed and the diffusion controlled combustion is changing and at the end of the transient when the start of injection was retarded for close to stationary operation. For the parts of the transient where the start of injection was held constant, an averaged heat release rate for each part was used.

To achieve a good correlation between measured and simulated data with respect to pressures and turbocharger speed the turbocharger performance data had to be adjusted. During the transient, only the turbine performance data were adjusted. The size of the turbines had to be adjusted in two steps due to the different VGT positions with respect to pressures in the gas exchange system. Efficiencies were then manually adjusted during the transient to achieve a good match between simulated and measured turbocharger speeds. The EGR valves were held open during the entire run and the VGT position was adjusted to achieve 30% EGR before and after the load demand. During the transient the VGT was held open, to get a maximum expansion of the exhaust gases over the high pressure turbine, which was assumed to give the fastest pressure build up.

Comparison of the simulated engine performance parameters, which influence the turbocharger performance, are shown in Figure 6.21, i.e. with respect to turbocharger speeds, exhaust pressures and inlet mass flow rate air. The engine simulations capture the engine behaviour. However, the engine performance parameters were somewhat adjusted to get closer to the measurements. The size of the turbochargers had to be adjusted during the simulation to give satisfactory results with respect to intake and exhaust pressures and turbocharger speeds. At low load the compressors were made 15% smaller and the low pressure turbine 10% larger than from the extrapolated turbine performance maps to achieve correct transient starting conditions. During the transient only the low pressure turbine was adjusted which was to make it 10% larger. At high load when the VGT was closed both the high pressure turbine and the low pressure turbine were made 10% smaller compared to the measured performance turbine maps. The turbine efficiencies had to be adjusted from the start of the transient to obtain a good match between simulated and measured turbocharger speeds, up to +5% for the high pressure turbine and between -8 to +2 % for the low pressure turbine depending on the VGT position.

The simulated temperature traces before and after the turbines, Figure 6.22, shows a different behaviour than the measured traces, which is due to the slow response time of the thermocouples used. A measurement was also performed with a 3mm K-type metal sheathed thermocouple to show the impact of its different sizes.
6.3. SIMULATION OF A TWO-STAGE TURBOCHARGED ENGINE

Figure 6.21. Measured and simulated turbocharger speeds, upper, mass flow rate of air, mid and cycle-averaged pressures upstream and downstream the HP-turbine and downstream the LP-turbine, lower figure.
6.1D ENGINE SIMULATION RESULTS

Figure 6.22. Measured and simulated temperature upstream and downstream the high pressure turbine and downstream the low pressure turbine. Measurements performed with 1.5 and a 3mm in outer diameter metal sheathed K-type thermocouples.

The EGR rate, from the simulation results during the transient, which could not be measured, is shown in Figure 6.23. At the start of the transient there is a EGR spike due to the sudden increase in back pressure when more fuel is injected. After the spike, the EGR rate decreases due to the lowered pressure difference across the engine. As the turbocharger speeds increases, the turbocharger efficiencies will increase (at least to a certain point) and hence, the pressure ratio across the engine drops.

6.3.3. VGT Position for Rapid Transient Response

A parameter study was performed to show the effect of different VGT positions on the transient response, which changes the balance of the expansion of the exhaust gases across the two stages. The calibrated engine model was used with the EGR valves closed, to not get an affect by different EGR rates, and
run with four different VGT positions chosen arbitrary. The simulation results are compared to measured data for the case with a fully open VGT, denoted 100%.

Figure 6.24 shows mass flow rate air, indicated mean effective pressure (IMEP) and the pressure before the HP-turbine at different VGT positions during the transient. With respect to the mass flow rate of air, the best solution is to close the VGT to 41%. Considering the IMEP build up, the transient response performance gets worse by closing the VGT as much as to 41% open. The best solution is therefore, with respect to transient response and mass flow rate air of the VGT positions tested to close the VGT to 66%. This gives similar response as with the VGT fully open, but with a higher mass flow rate air. This makes it possible to run with a higher rate of transient EGR without losing in transient response and air to fuel ratio. As expected, the back pressure rises the more the VGT is closed. However

The interaction between the two turbochargers is shown in Figure 6.25, which shows the turbocharger speeds for different VGT positions. The more the VGT is closed the more of the exhaust gases will expand over the VGT and the LP-turbocharger speed will raise while the HP-turbo speed will decrease.
Figure 6.24. Mass flow rate of air, upper, IMEP mid and pressure upstream the HP-turbine for different VGT positions, lower figure. Measured data for fully open VGT is included.
6.3. SIMULATION OF A TWO-STAGE TURBOCHARGED ENGINE

Figure 6.25. Turbocharger speeds, HP-turbo, left and LP-turbo at different VGT positions, right. 100% denotes fully open. Measured data for fully open VGT is included.

6.3.4. Discussion

The results show that adjusting the measured turbine performance data with up to 10% was needed to achieve a good match between simulation results and measurements. The turbine performance data were adjusted with respect to both turbine size and efficiency. There are several reasons why adjustments of the turbocharger performance data might be needed even though, in this case some of the adjustments were not necessary to the whole engine performance since some of the errors in performance cancel out each other.

- For engine modelling at low loads the performance data have to be extrapolated, which is uncertain due to the non-linear behaviour of the turbine and compressor.
- The turbine model does not model the twin-entry volute physically correct due to that cross flows within the turbine are simply modelled via a single orifice connection between the inlet pipes to the turbine object.
- The turbine performance map for the twin-entry turbine is measured under equal admission conditions, which will not represent the engine performance at partial admission conditions as shown in Section 6.1.
- The exhaust manifolds upstream the turbines are not straight on the engine. For the high pressure turbine there is a double bend similar to the one discussed in Section 7.1, which introduces a swirling motion. The high-pressure turbine will give a swirling motion to the flow, which
flows through a 90 degree bend before reaching the low pressure turbine. The bend will give a non-uniform flow with a possible separation zone at the inner radius downstream the bend. These secondary flows are not considered by the 1D code or by turbocharger performance maps used.

- The turbine maps are used under the assumption that the turbine works as a quasi-steady flow device even though the pulse amplitude into the HP-turbine is relatively high.

The pressure trace before the high pressure turbine, Figure 6.17 is less well matched at the crank-angle intervals where cross-flow within the turbine occur for the simulation with the EGR valves open. The reason for this can be that the flow is more disturbed since the outlet to the EGR-route is located close to the turbine inlet. This can make the simple twin-entry turbine model not good enough or due to that the complex flow field within the exhaust collector, which consists of a four-way split that has a 3D geometry cannot be accurately modelled in 1D. The lower simulated peak pressures before the high pressure turbine compared to measured peak pressures could not be explained. This can imply that the simulated energy to the turbine is lower than on the engine, which implies that the raised turbine efficiency is needed to compensate for the lower energy. The energy into the turbines could not be measured due to difficulties to measure instantaneous gas temperature and mass flow rate.

Figure 6.19 shows that there is a discrepancy both with respect to amplitude and phasing of the turbocharger acceleration (phasing mainly for the LP turbo). As explained earlier, the volute is modelled via a straight pipe to account for the volume of the volute, which the turbine maps do not take into account. The volume of the volute, especially for the low pressure turbine did affect the pressure amplitude after the high pressure turbine significantly. The volute is not a straight pipe as modelled, which will affect the distance for the flow to pass before hitting the rotor.
CHAPTER 7

Results for Reduced Models based on LES

The following sections will show results from the two methods to derive reduced models for engine simulation purposes, described in Section 5. The methods enable a reduction of the Navier-Stokes equations in a systematic way based on LES.

The first method presented is a length scale analysis, shown for a double bent pipe. The method allows a quantification of the terms in the Navier-Stokes equations projected onto a flow path, in order enable a reduction of the non-dominating terms. The second method is a Galerkin projection of the Navier-Stokes equations onto POD modes, which is based on LES results for the double bent pipe with a stationary inlet condition. Furthermore, the POD modes are computed for the double bent pipe with a pulsating inflow condition, for a radial turbine and a ducted orifice plate, to show its properties and the possibilities to obtain a reduced model.

The LES results for the double bent pipe were computed within this work, which results from the 1D simulation approach are compared to, to show its behaviour for a case with secondary flows. The results for the radial turbine were computed by Hellström & Fuchs (2009) and for the circular duct with an orifice plate by Alenius et al. (2011).

7.1. Flow in a Double Bent Pipe

To show the concepts used to reduce the Navier-Stokes equations, the simplest geometry used is a double bent pipe. The pipe consist of two bends, curved in different directions to obtain a flow field that can not directly be seen as a 1D or 2D flow case, Figure 7.1. The double bent pipe has a diameter of 10 mm. The radiuses of the bends are 10 mm and 15 mm, respectively. The straight parts of the pipe are; five pipe diameters upstream the first bend, two diameters downstream the first bend and 10 diameters downstream the second bend. The mesh has 1956000 cells, with 2400 cells in each cross section, which gives a characteristic cell length of 0.2 mm. Five diameters after the second bend the cells are stretched 5%. The simulation is performed with an inflow velocity of 50 m/s with a top-hat profile and a temperature of 400 K, which corresponds to a Reynolds number of 20000. The time-averaged flow field and
a snapshot of the instantaneous flow field are shown in Figure 7.2 to visually exemplify the flow in a pipe system.

The length scale analysis will basically give a measure of the secondary flow components present in the flow field, i.e. flow components in the spanwise direction in relation to the flow path. The secondary flow components are seen by studying the in-plane velocities downstream the bends, Figure 7.3. After the first bend, two counter rotating vortices appear. The origin of these vortices is the skewed velocity profile starting to form in the bend, due to centrifugal forces. This will give a high pressure at the inner radius, where a separation zone is formed, and a lower pressure at the outer radius, where there is a high flow velocity driving the in-plane flow towards the outer part of the pipe. After the second bend, the flow forms a swirling motion, since the flow, which now has a skewed velocity profile, is deflected in the second bend of the pipe.

To show the scales of the turbulence that are resolved with the LES, the power spectral density (PSD) is computed for different probe locations, which shows the power distribution as a function of frequency. Figure 7.4 shows the PSD for the axial velocity component for probes located one, three and five diameters downstream the second bend in the center of the pipe and for probes located in a plane one diameter downstream the second bend. The results show that the inertial subrange is captured almost over an order of magnitude of the frequency spectrum, down to $10^4$ Hz. The PSD computed for different radially positioned probes shows that the fluctuations are stronger in the inner part of the pipe than in the outer region.
7.1. FLOW IN A DOUBLE BENT PIPE

7.1.1. Pulsating Inflow Condition

The flow through the double bent pipe with a pulsating inflow condition was simulated to show the flow structures for different POD modes for engine like conditions. The inlet flow velocity for the simulation is based on an engine simulation of a passenger car engine. The pulse period was shortened and its frequency increased from 30 to 200 Hz to shorten the computational time. To prevent back flows the minimum velocity was set to 10 m/s and to decrease the velocity gradients the velocity was lowered by a factor of 2. The inlet temperature was set to 800 K, which gives a speed of sound of 545 m/s. The flow conditions correspond to a Reynolds number of 1200 to 27000 using the pipe diameter as the characteristic length. The velocity at two probes located in the centre of the pipe, one diameter downstream of the inlet and four diameters downstream the second bend, respectively, are shown in Figure 7.5. The velocity correlation for these two probe locations shows that the velocity traces are phase shifted by $2.4 \times 10^{-4}$ s, i.e. 0.05 periods, which corresponds to a propagation velocity of 584 m/s. The propagation velocity based on the probe one diameter downstream of the inlet and eight probes located at different positions, closer to the inlet than the second probe gives a propagation velocity of 526 to 591 m/s, without any direct trend. Hence, the discrepancy from the
7. RESULTS FOR REDUCED MODELS BASED ON LES

Figure 7.3. Planes after a half pipe diameter downstream the first bend, left and downstream the second bend, right. Arrows shows the in-plane velocities scaled accordingly and colour shows the magnitude of the time-averaged velocity field.

Figure 7.4. Power spectral density for probes located 1, 3 and 5 diameters downstream the second bend, left and probes located in a plane 1 diameter downstream the second bend, right.

speed of sound could be due to that the pulse propagation speed is influenced by the bulk velocity or due to numerical errors.
7.1. FLOW IN A DOUBLE BENT PIPE

7.1.2. The 1D Approach Compared to LES

A comparison of the simulation results from the 1D approach with results from LES are performed to show to which extent the 1D approach can capture the pressure drop along a pipe system where secondary flow effects are expected. The LES results were space-averaged at each cross-section, normal to the central streamline, Figure 7.6, and along the geometrical centre line. The corresponding space-averaged pressure fields are depicted in Figure 7.7.

Comparing the LES results with the 1D computation shows that the pressure drop is similar for the two simulation strategies upstream the second bend, i.e. until approximately 9 pipe diameters from the inlet of the pipe. However, in the second bend and downstream the second bend the pressure drop differs significantly. The 1D simulation approach gives the same pressure drop for all the three straight parts in the pipe while the LES computation shows clear differences due to upstream effects, which are not taken into account by the 1D approach. The flow will be less distorted the further away from the bend it is observed where the pressure gradient is decreasing at first. The mean pressure gradient by the 1D model has a considerably larger value as compared to the LES result. The slope of the 1D simulation is almost three times as large as the corresponding LES result at the outlet of the pipe (i.e. between 12-20D downstream). To account for this deviation, the 1D model has to be re-calibrated (in terms of pressure loss). The plane averaging according to

![Velocity trace from a probe located one pipe diameter downstream the inlet and four pipe diameter downstream the second bend.](image-url)
7. RESULTS FOR REDUCED MODELS BASED ON LES

Figure 7.6. Central streamline of the double bent pipe with an inlet flow velocity of 50 m/s.

Figure 7.7. Time averaged pressure from a LES averaged at each cross section with a normal direction according to the central stream and the geometrical centre line compared to 1D simulation results. The central streamline or the geometrical centre line does not show significant differences for the plane mean-pressure distribution.
7.2. The Radial Turbine

Flow data of a radial turbine was used to analyse the energy distribution of its POD modes. The turbine used for the computations is a nozzle-less 9 bladed radial turbine where the leading edge tip radius of the wheel is 22.1 mm and the trailing edge tip radius is 18.75 mm. The inflow velocity was set to 180 m/s, with an inlet temperature of 873 K and a rotational speed of 97897 rpm.

A comparison of the flow data computed via LES and the 1D approach could not be performed, since the flow through the turbine is not computed by the 1D approach used for the engine simulations. Instead the pressure and velocity along two particle-paths, Figure 7.8, are extracted to show how the incoming flow is distributed around the rotor. The particle-paths are computed by following a mass-less particle by integrating the velocity vector in time. As seen, the two tracked particles pass through different pairs of blades and exhibit a strong deviation from a particle-path in a straight pipe.

Following the two particle-paths, shows that the inflow at different inlet positions to the turbine will reach the turbine rotor at different azimuthal angles. Studying the velocity and pressure along the particle-paths, Figure 7.9, shows that the velocity and pressure around the circumference are similar at the positions where it reaches the rotor inlet. However, the flow will reach the rotor at different time instances. The same will hold for a pulsating inflow condition, where the pulse will reach the rotor inlet at different time instances around its circumference and be mixed by other parts of the pulse. This behaviour have to be captured by a low-order turbine model to have correct inflow conditions at the turbine rotor.
7.3. Length Scale Analysis

Next, we consider the reduction of the 3D flow by studying the possibility of re-scaling the governing equations. The terms of the incompressible Navier-Stokes equations, Equation 5.15, projected along the flow path of the geometry will be quantified to enable a reduction of the non-dominating terms. For the Navier-Stokes equations to be valid after the projection, the equations have to be based on a curvilinear coordinate system according to Equation 5.17 and 5.29. Hence, in order to compute the terms, including the scale factors, the flow fields have to be transformed onto a domain with the chosen curvilinear coordinate system, i.e. onto the so called computational domain.

The convective and viscous terms of the momentum equations are of main interest. The pressure terms are not quantified since they need to be kept for a reduced model to balance out the equations. Since we are primarily aiming at quantifying the effects from the secondary flow components, we will only consider the momentum equation in the axial direction, i.e. in $e_S$. The length scale analysis is computed for the double bent pipe, but could be computed
7.3. LENGTH SCALE ANALYSIS

Figure 7.10. Specific kinetic energy of the flow computed from the projected velocity field onto the geometrical centreline and the central streamline averaged at each cross section of the pipe. The dashed vertical lines show the location of the bends. $D$ denotes pipe diameters.

in a similar manner for the radial turbine based on a chosen particle-path and using a single blade passage that the particle passes through. As a second step, a reduced model can be derived by neglecting the non-dominating terms for the specific geometry.

The transformation can be computed according to the geometrical centreline or according to e.g. the mean-streamline starting at the centre of the inlet. The kinetic energy of the time-averaged velocity field projected onto the geometrical centreline and the central streamline for the double bent pipe is shown in Figure 7.10. The results show that projecting the flow field onto the geometrical centreline captures more of the kinetic energy than a projection onto the central streamline. This is mainly due to the swirling motion, shown in Figure 7.3, after the second bend encountered by the streamline, which is off angle with respect to the bulk of the flow. The geometrical centreline was therefore chosen for the projection of the Navier-Stokes equations.

The coordinates for the transformation are $(S, \eta, \xi)$, where $S$ follows the geometrical centreline of the geometry, $\eta$ is in the "radial" direction and $\xi$ is in the "azimuthal" direction. The transformation can basically be seen as the geometry is being cut open according to Figure 7.11 at the discontinuity in the azimuthal direction and unfolded. The cut will correspond to the side walls of the computational domain, a channel in this case. The inner radius, a small distance from the centre of the pipe, which is not included in
the transformation, will be transformed onto the lower wall of the channel. The cylinder walls are transformed onto the upper wall. The transformation is depicted in Figure 7.12, which shows the magnitude of the time-averaged velocity field in the $S-\eta$-plane in the computational domain, at $\xi = \pi/2$ and $\xi = 3\pi/2$, and corresponding planes in the physical domain, i.e. the double bent pipe. To see the continuity of the velocity field on the two planes from the channel is difficult in the part close to the first bend due to that the central geometrical line is not included in the transformation, which is represented by the white field, and due to the strong velocity gradients at the centre of the pipe.

The magnitude of the convective and viscous terms for each component computed with respect to the curvilinear coordinates are presented in Figure 7.13 as area-weighted averages across the axial direction. The terms are computed as time-averages based on 100 snapshots of the flow field with a time step of $2 \cdot 10^{-4}$ s.

Comparing the viscous terms for each component shows that the viscous term in the radial direction, with respect to the geometrical centreline, is approximately two orders of magnitude larger than the viscous term in the axial and azimuthal direction, respectively. This is due to that the radial component is large close to the boundary where the velocity gradient is large due to the no slip condition at the wall. The viscous term in the radial direction close to the wall is $2 \cdot 10^5 m/s^2$, locally. This is in accordance with the derivation of the boundary-layer equations for a flow over a flat plate, where the viscous term in the stream wise direction is small compared to the wall normal direction. Hence, considering the loss terms only indicates that the viscous terms in the axial and in the azimuthal direction can be neglected without large discrepancies. Worth noting is that the viscous term in the radial direction is largest.
just after the exit of the bends due to a relatively strong shear layer close to the separation bubble.

Comparing the convection of the different velocity components shows that they are all in the same order of magnitude with respect to the geometrical centre line. This is also in accordance with the derivation of the boundary-layer equations where the convective terms in the stream wise and wall normal direction are of the same order of magnitude. The results show that the convection is in general large in the bends with an increase upstream of each bend and a recuperation zone downstream of each bend. Hence, secondary flow effects from the bends will affect the flow field both up and downstream of the bends and the flow fields within the bends will therefore be affected by each other. It is also noticed that the flow is dominated by the inertia effects, while viscous effects are small altogether. The ratio between the convection of the radial and axial velocity component and the azimuthal and axial velocity component are shown in Figure 7.14. The results show that the convection of the radial velocity component is relatively large within the bends in relation to the convection of the axial velocity component, which is due to the centrifugal forces that the flow encounters.
Figure 7.13. Magnitude of the viscous terms in curvilinear coordinates, left and the convective terms, right, area-weighted averages across the axial direction according to the geometrical centreline. The figures in the vertical directions are for the "S", "η" and "ξ" directions, respectively. The dashed vertical lines show the locations of the bends.
Comparing the convection of the azimuthal velocity component to the convection of the axial velocity component shows that the convection of the azimuthal velocity component is relatively large after the second bend, which is due to the swirling motion encountered by the flow. This implies that neglecting, e.g., the terms related to the azimuthal direction, which would result in a 2D model, will give a relatively large error especially after a second bend with a skewed inflow velocity profile, due to the strong swirling motion that the model will not be able to capture.

7.4. Galerkin Projection onto POD Modes

The concept of deriving a reduced model via Galerkin projection onto POD modes for internal flows of engine components is given in this section. The properties of the POD will first be shown for three different flow geometries; a ducted orifice plate, a double bent pipe with a stationary and a pulsating inflow condition and a radial turbine.

The ducted orifice plate is shown due to its strong coherent structures that can be detected by reconstructing the velocity field by a small number of POD modes. The double bent pipe is used to show how a pulsating inflow condition will affect the flow structures in the flow field compared to a stationary inflow condition, which is of significant importance for the engine application. The POD is also computed for a radial turbine, to discuss how the concept can be used for this type of flow device.

7.4.1. POD for a Ducted Orifice Plate

The orifice plate is placed within a circular duct and has a centrally located orifice with an area contraction ratio of 0.28. The inflow velocity to the duct
is 27.5 m/s giving an inlet Mach number of 0.08. The POD is computed from 480 snapshots sampled every $2.5 \cdot 10^{-3}$ s.

Reconstructing an instantaneous flow field with a truncated set of POD modes filters away the low-energy content flow structures. For a stationary flow case the zero-th mode, which is the structure that contains the most of the flow energy, will represent the time-averaged flow field. This can be seen in Figure 7.15, which shows the time averaged velocity field in the duct and the velocity field reconstructed from the zero-th POD mode.

The time coefficient used for the reconstruction of the velocity field with only the zero-th mode is insignificant, since it corresponds to the time-averaged field, and hence is time-independent. To show the properties of the POD, flow fields were reconstructed with different amount of POD modes and compared to an instantaneous flow field.

Figure 7.16 shows an instantaneous flow field and the corresponding isolines of the $\lambda_2$-criterion, which detects the vortex cores. Figure 7.17 and 7.18 shows the isolines of the $\lambda_2$-criterion computed for reconstructed velocity fields using the first 50 and 10 POD modes, respectively. The reconstructed velocity fields correspond to 94.8 % and 92.7% of the energy content of the flow field. As the low-energy containing flow structures are filtered out, due to the truncation of the POD modes used in the reconstruction, coherent structures that are difficult to find in an instantaneous flow field can be seen as the ring-vortices that roll up after the orifice plate.
7.4. GALERKIN PROJECTION ONTO POD MODES

Figure 7.16. Left figure, instantaneous velocity field. Right figure, isolines of the $\lambda_2$-criterion for the instantaneous velocity field.

Figure 7.17. Isolines of the $\lambda_2$-criterion for the reconstructed velocity field from the first 50 POD modes containing 94.8\% of the kinetic energy of the flow field.

Figure 7.18. Isolines of the $\lambda_2$-criterion for the reconstructed velocity field from the first 10 POD modes containing 92.7\% of the kinetic energy of the flow field.
7.4.2. POD for the Double Bent Pipe

The results for the POD computed for the double bent pipe for a flow case with a stationary and a pulsating inflow condition will be shown. For the POD computations the pipe was restricted, just upstream the first bend and in the middle of the straight pipe part downstream the second bend, to perform the computations on the parts where the most interesting flow phenomena are found. The case setup is discussed in Section 7.1.

The final POD for the flow field with the stationary inflow condition was computed from 780 snapshots. The sampling rate was chosen after computing the autocorrelation for the velocity components at different probe locations, which gives a time limit for two samples being uncorrelated. The autocorrelation for the two probe locations are shown in Figure 7.19, which gave a time limit of $2 \cdot 10^{-4}$ s.

As a first test the POD was computed with different amount of snapshots to see its effect on the energy distribution among the modes. Figure 7.20 shows the energy and cumulative energy distribution between the first eight modes. The results show that the energy distribution does not change significantly when more than 400 snapshots are used.

The higher-order time coefficients should correspond to coherent structures and evolve in time as they do. However, since the zero-th POD mode correspond to 96.7 % of the energy of the flow field the higher-order modes will not contain any strong coherent structures.

The time coefficients, which scales the flow structures for each mode in time, for the x-component of the first to third POD modes are shown in Figure 7.21. From the first to the third time coefficient shown, it can be pointed out that no clear dependency can be seen between the samples, which imply that
7.4. GALERKIN PROJECTION ONTO POD MODES

<table>
<thead>
<tr>
<th>No. Snapshots</th>
<th>Energy/ Total Energy</th>
<th>Cumulative Energy/ Total Energy</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>0.966</td>
<td>0.966</td>
</tr>
<tr>
<td>200</td>
<td>0.967</td>
<td>0.967</td>
</tr>
<tr>
<td>300</td>
<td>0.968</td>
<td>0.968</td>
</tr>
<tr>
<td>400</td>
<td>0.969</td>
<td>0.969</td>
</tr>
<tr>
<td>500</td>
<td>0.970</td>
<td>0.970</td>
</tr>
<tr>
<td>600</td>
<td>0.971</td>
<td>0.971</td>
</tr>
<tr>
<td>700</td>
<td>0.972</td>
<td>0.972</td>
</tr>
<tr>
<td>800</td>
<td>0.973</td>
<td>0.973</td>
</tr>
<tr>
<td>900</td>
<td>0.974</td>
<td>0.974</td>
</tr>
</tbody>
</table>

Figure 7.20. Energy distribution for the first modes computed for different number of snapshots. Normalized energy distribution per mode for mode 1-7, left and cumulative energies for mode 0-7, right.

Figure 7.21. Time coefficients for the inlet axial velocity component for the first to third POD modes of the double bent pipe with stationary inflow condition.

The snapshots taken are statistically independent. The zero-th time coefficient is constant in time and is related to the time averaged flow field as for the case with the ducted orifice plate.

For the pulsating flow case with the inflow velocity profile according to Figure 7.5, the POD modes were computed from 1000 snapshots over 28 pulses with 40 samples per pulse period. The results show that the zero-th mode for the case with a pulsating inflow condition contain 96.7% of the kinetic energy.
of the flow, i.e. as for the case with a stationary inflow condition. However, for the pulsating flow case, the first modes that represents the coherent structures, contain more energy than for the case with a stationary inflow condition, Figure 7.22. For the case with the pulsating inflow condition, the zero-th time coefficients, shown in Figure 7.23, varies in time in a similar manner as the pulses. These two results imply that the zero-th flow structure represents the flow field for a large range of flow velocities, i.e. at least from 10 to 220 m/s, which are the minimum and maximum inflow velocities during the inflow pulse.

The results imply that a reduced-order model could be based solely on the zero-th POD mode, which could be used for a large range of flow conditions simply by scaling the zero-th flow structure according to changes in inflow condition. However, velocity fields reconstructed from 50 POD modes at the minimum and maximum velocities during a pulse shown downstream the first bend, Figure 7.24, shows that the high-order modes can still locally capture flow structures that can be of significant importance for the flow behaviour, as e.g. the size of the separation bubble downstream the first bend. Studying the first and second time coefficients, Figure 7.23, shows peaks for the first POD mode at the positive slopes of the pulses and for the second POD mode at the negative slopes. This implies that there is a hysteresis effect throughout the pulse. Hence, the flow field can not be seen as being completely quasi-steady. However, this should give a relatively small effect to quantities as the pressure drop since the zero-th POD mode contains the main energy content.

**Figure 7.22.** Energy content for different number of modes for the stationary and pulsating inflow conditions.
7.4. GALERKIN PROJECTION ONTO POD MODES

Figure 7.23. Zero-th time coefficient in the left figure for the three velocity components and first to third time coefficient in the right figure for the inlet axial velocity component computed for the double bended pipe with pulsating inflow condition.

Figure 7.24. Magnitude of the reconstructed flow field using 50 POD modes downstream the first bend, low and high velocity snapshots during pulsating flow conditions.

7.4.3. POD for a Radial Turbine

The POD modes are computed for the radial turbine to see the energy distribution between the modes and the possibilities to perform a reduced model via the Galerkin projection of the Navier-Stokes equations onto the POD modes. The POD modes are computed from 240 snapshots sampled every $2 \cdot 10^{-5}$ s, i.e.
approximately 30 samples per wheel revolution. For this case the inlet velocity was set to 116 m/s with a temperature of 1273 K, which corresponds to a mass flow of 0.081 kg/s.

The POD mode analysis shows that the zero-th mode contains 69.7 % of the kinetic energy and the first and the second modes contain 13.2 and 12.9 %, respectively. The energy content for the remaining modes is below 1 %, each. Reconstructed velocity fields for the different modes are shown in Figure 7.25, which clearly shows that the energy for the first and second modes are mainly located in the turbine rotor. The velocity field reconstructed from the first 50 POD modes shows that the flow field in the rotor and the volute are not aligned, which has to be considered for the Galerkin projection. The corresponding time coefficients are shown in Figure 7.26 and Figure 7.27. The zero-th time coefficients show eight sharp peaks with the same frequency as the rotation of the wheel. Hence, the zero-th structure is influenced by the rotation of the wheel. The first- and second- time coefficients show a clear harmonic motion where the two signals are 90 degrees out of phase, due to orthogonality. To consider wave propagation, i.e. that the POD modes would represent a wave-like periodic structure or a periodic motion, two time coefficients are expected to be in pairs with similar amplitudes, but out of phase. This is the case for the first and second time coefficients, which consider the wheel rotation. The left figure, in Figure 7.27, shows the y-component, which is in the radial direction to the turbine wheel while the right figure shows the z-component, which is in the direction parallel to the turbine axle. The amplitude of the radial time coefficients are more affected by the wheel rotation than the axial component, as expected.
Figure 7.25. Reconstructed velocity fields, zero-th POD mode, upper left containing 69.7% of the energy of the flow field, first mode, upper right containing 13.2%, second mode, lower left containing 12.9% and lower right figure shows the reconstructed velocity field from the first 50 modes.
7. RESULTS FOR REDUCED MODELS BASED ON LES

7.4.4. Galerkin Projection for the Double Bent Pipe

The Galerkin projection is demonstrated in this section computed for the double bent pipe. The non-linear coupled time dependent system of ODEs obtained through Galerkin projection have been computed for the fluctuating velocity field using 10, 20 and 40 POD modes, respectively. The time marching of the system of ODEs is performed with the classical fourth-order Runge-Kutta method.

A first solution computed with a time step of $1 \cdot 10^{-6}$ s shows that the system is unstable and that the solution blows up after only $2 \cdot 10^{-3}$ s. However,
freezing the second convective term, \(-\sum_{i=1}^{N} \alpha_i^j \left( \varphi_i^j \frac{\partial u_i}{\partial x_j}, \varphi_k^l \right)\), makes the system over damped. Hence, this implies that the second convective term, which is linear, has a large influence on the stability of the system. A damping to the linear terms was therefore added by setting the positive real parts of the linear system’s eigen-values to zero.

Erroneous results can be encountered if the time step used to solve the system of ODEs is too large. Time marching with different time steps was therefore performed for the non-linear system of ODEs using 10 modes, including the damping of the linear terms. Figure 7.28 shows the solution obtained with four different time steps from \(1 \cdot 10^{-6}\) to \(1 \cdot 10^{-4}\) s. The results are similar for the two cases computed with a time step of \(1 \cdot 10^{-6}\) and \(1 \cdot 10^{-5}\) s. The solution obtained with larger time steps gives a solution with lower amplitudes, where the solution computed with a time step of \(1 \cdot 10^{-4}\) s gives a clearly damped solution, hence influencing the numerical solver. This implies that a time step below \(1 \cdot 10^{-5}\) s is needed to compute the actual solution to this system of ODEs.

Results for the time marching of the non-linear system of ODEs including damping of the linear terms is shown in Figure 7.29, including the time coefficients from computing the POD. The results are computed for 3 different sets of ODEs derived with 10, 20 and 40 POD modes. The results show that the solution becomes unstable even though damping is added to the linear terms. The more POD modes that the model is based on, the sooner will the solution get unstable. For the case with 10 modes the solution to the system of ODEs goes unstable after approximately 0.1 s. However, the solution to the system

**Figure 7.28.** Effect of different time steps on the solution of the non-linear coupled system of ODEs using 10 modes.
Figure 7.29. Solution of the non-linear system of ODEs including stabilisation of the linear terms using 10, 20 and 40 POD modes. The case with 10 modes gets unstable after 0.1 s.

Based on 10 modes shows that it is in the same order as the time coefficients computed for the POD.

Figure 7.30 shows the solution to the system of ODEs, compared to two of the time coefficients obtained through computation of the POD for the first part of the time marching. The results show that the model is not able to predict the time coefficients accurately. The discrepancy is believed to depend on the fact that the fluctuating flow field does not contain any strong coherent structures, which is the basic property of the approach.

The relative errors for the reconstructed fluctuating velocity field are computed for the linear terms \((f(\cdot))\) using a finite number of POD modes. The relative errors are computed as,

\[
\frac{\sum (f(u_i) - f(\phi_i))^2}{\sum f(u_i)^2}
\]

where the resulting relative errors are presented in Table 3. The results show that the terms are not well presented at all, even though the terms are better represented with an increased number of POD modes. This supports the fact that it is not feasible to re-create incoherent structures such as turbulence, with this approach.

To assess the influence of the non-linear part on the solution to the system of ODEs, the time marching was computed with and without the non-linear
Figure 7.30. Solution of the non-linear system of ODEs including linear stabilisation for 10 and 40 modes compared to the time coefficients from the computed POD.

Table 3. Relative error for the reconstructed fluctuating velocity field and the linear terms using 10, 20, 40 and 70 modes for the double bent pipe with a stationary inflow condition.

<table>
<thead>
<tr>
<th>no. modes</th>
<th>$\sum_{l=1}^{N} \alpha_l \phi_l$</th>
<th>$\sum_{l=1}^{N} \alpha_l u_j \frac{\partial \phi_l}{\partial x_j}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>(0.94, 0.93, 0.91)</td>
<td>(0.99, 0.99, 0.99)</td>
</tr>
<tr>
<td>20</td>
<td>(0.85, 0.87, 0.82)</td>
<td>(0.98, 0.98, 0.97)</td>
</tr>
<tr>
<td>40</td>
<td>(0.80, 0.82, 0.79)</td>
<td>(0.96, 0.96, 0.96)</td>
</tr>
<tr>
<td>70</td>
<td>(0.76, 0.77, 0.72)</td>
<td>(0.94, 0.94, 0.93)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>no. modes</th>
<th>$\frac{\sum_{l=1}^{N} \alpha_l \phi_l}{\mu \sum_{l=1}^{N} \alpha_l \phi_l}$</th>
<th>$\frac{\mu}{\frac{\partial}{\partial x_j} \sum_{l=1}^{N} \alpha_l \phi_l \phi_l}{\frac{\partial}{\partial x_j} \sum_{l=1}^{N} \alpha_l \phi_l \phi_l}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>(0.98, 0.98, 0.98)</td>
<td>(0.99, 0.98, 0.96)</td>
</tr>
<tr>
<td>20</td>
<td>(0.95, 0.97, 0.95)</td>
<td>(0.97, 0.96, 0.93)</td>
</tr>
<tr>
<td>40</td>
<td>(0.92, 0.94, 0.93)</td>
<td>(0.94, 0.95, 0.92)</td>
</tr>
<tr>
<td>70</td>
<td>(0.87, 0.90, 0.91)</td>
<td>(0.93, 0.92, 0.90)</td>
</tr>
</tbody>
</table>

terms using 10 and 40 POD modes, respectively. Figure 7.31. Using 10 modes, the non-linear terms does not give a significant effect on the stable part of the solution. However, for the case using 40 modes, there is a clear discrepancy between the solutions of the linear and non-linear system of ODEs. The solution computed for the case including the non-linear terms using 40 modes is unstable and blows up after 0.01 s while the linear system is stable, as expected due to
the added damping. For a system based on a large number of POD modes, the flow field will contain larger velocity fluctuations. Hence, the non-linear terms will have a larger effect on the solution. This implies that there is a need for further damping if several POD modes are needed to obtain an accurate low-order model.

The PSD of the time signal for the x-component of the first time coefficient is shown in Figure 7.32. The PSD is performed on a time series of 5 seconds computed with the linear system of ODEs based on 10 and 40 modes. The results show that the solution obtained with the model that is based on 40 modes resolves more frequencies than the model based on 10 modes. However, it should be noted that the simulations are resolved to approximately $2 \cdot 10^4 \text{Hz}$ according to Figure 7.4. For other time coefficients the PSD looks similar, i.e. the peaks are located at similar frequencies.
7.4. GALERKIN PROJECTION ONTO POD MODES

Figure 7.32. The PSD for the solution of the \( x \)-component of the first time coefficient computed with the linear system based on 10 and 40 modes.

7.4.5. Discussion, Galerkin Projection for the Radial Turbine

The flow field for the radial turbine was computed with the LES approach using the sliding mesh technique to consider the rotation of the wheel. If the snapshots are taken with the rotor in different positions, the mesh will change in time and the different flow structures, as shown Figure 7.25 will give a discontinuity at the boundary between the turbine rotor and the volute. Hence, dividing the terms into a temporal and a spatial part will not be possible since the geometry will change in time. However, there are different possibilities to solve this issue,

1. Sample the snapshots with the rotor in the same position, e.g. to take nine samples at each revolution for the nine-bladed turbine rotor used.
2. To consider the turbine rotor rotation in the LES by the rotating reference frame technique, which considers the rotation of the rotor by added terms to the Navier-Stokes equations instead of actually rotating the rotor as with the sliding mesh technique.
CHAPTER 8

Summary, Conclusions and Outlook

The simulation tools used for engine design are based on the one-dimensional Euler equations. To account for losses, an added source term is used with the assumption that the flow is quasi-steady, even though, as has been shown for the turbine by several researchers, there is a clear discrepancy of the performance for steady and pulsating flow conditions. Another drawback is that secondary flow effects cannot be accounted for in the 1D framework, which has been found to be important in the IC-engine gas exchange system. A comparison of the simulation results for a double bent pipe computed with the 1D approach and LES has shown that the losses in a second bend differs significantly. The mean pressure gradient computed with the 1D model has a considerably larger value as compared to the LES results. To account for this deviation, the 1D model has to be re-calibrated in terms of pressure loss.

In addition to pressure loss, the 1D model yields a false inflow profile to e.g. the turbine, which leads to substantial errors in predicting the turbocharger behaviour in the system. Two approaches to derive reduced models in a rational way, based on LES, have therefore been investigated in this thesis. These approaches are aimed at producing engine simulation tools with improved accuracy together with high computational efficiency, as compared to existing 1D tools.

8.1. 1D Engine Simulations

The 1D engine simulations showed that the engine model, especially the turbine model, has to be calibrated to reflect the engine performance both for stationary and transient engine operation. However, it is difficult to know how large adjustments to the turbine model that are actually needed, since the prediction of the exhaust gas energy into the turbine is uncertain and difficult to measure, especially for transient engine operation. Hence, it is difficult to quantify the modelling errors since there is a large uncertainty in the turbine inflow boundary conditions.

The calibrated engine model was used to compute the instantaneous turbine efficiency for a twin-entry turbine from a mix of measured and simulated data with the following results,
There is a significant asymmetry in the on-engine twin-entry turbine behaviour with a higher efficiency obtained for the flow from cylinders connected to the outer volute entry, which is in agreement to gas-stand measurements performed by Dale & Watson (1986).

Significant differences in turbine efficiency are shown for the flow from the outer cylinders compared to the inner cylinders, which are connected to the turbine with a shorter manifold.

8.2. Length Scale Analysis

A scaling analysis implies that one uses known information about the character of the flow (i.e. length and velocity scales) to assess the importance of different terms in the governing equations. By neglecting terms of least importance, or by maintaining terms that are of most importance, one may reduce the original set of equations in a rational manner. The approach quantifies the terms of the Navier-Stokes equations after changing the coordinate system with one coordinate aligned with the flow path, i.e. either the geometrical centreline or the streamline. The other directions can be defined to be orthogonal to the direction of the flow path and to each other (to simplify the transformation process). One may estimate the importance of the different terms due to gradients in the direction of the flow path and its span wise directions. Model reduction is attained by neglecting non-dominating terms. In fact, one may recover the 1D model if the flow is aligned with the pipe axis and when the curvature of the pipe is so mild that the gradients of the flow in the cross-plane are negligible. This approach has been applied to the flow in a double bent pipe using a flow field computed with LES. The computations for this case showed that,

- The radial component of the viscous term is two order of magnitudes larger than the direction of the axial and azimuthal component, which is in analogy to the boundary-layer approximation by Prandtl.
- The convective terms are all of the same order of magnitude. However, the radial component is of significant importance in the bends where the flow is affected by centrifugal forces. The azimuthal component is only significant after the second bend due to the swirling motion of the flow.

The approach can be applied to any internal flow situation including the turbine, if the flow situation allows for a reduction. However, for a turbine model it is of importance to be able to compute its power output, which cannot be done directly with high accuracy, unless the 3D flow field is computed.

The approach must be checked a posteriori to uncover possible inconsistencies. That is, if the computed results of the reduced problem show that the assumptions made for the reduction are violated. The approach could be
extended to be used adaptively, within a full 3D model, by using the simplified equations at regions where these are valid and keeping the full 3D approach where no reduction is possible. This approach is close in concept with what some researchers tried to do in combining 1D and 3D tools (Onorati et al. (2010)). The main difference in the proposed approach here is that the proposed combination is done on a rational basis with the possibility of an a priori and an a posteriori error estimation. Additionally, the approach would not suffer from the problem of the combined 1D-3D approach that is related to obtaining boundary conditions from the 1D solution. However, since the method has been applied to a single flow case only, i.e. the double bent pipe, the method should be tested for more complex geometries to show its potential for industrial applications.

8.3. Galerkin Projection onto POD Modes

The POD decomposes the flow field into a temporal and a spatial part, i.e. to the so called POD modes. To obtain a reduced model, the POD modes have been used as a basis for the Galerkin projection of the Navier-Stokes equations. The POD modes were computed for a double bent pipe, a ducted orifice plate and a radial turbine with the following results,

- The zero-th POD mode for the flow field of the double bent pipe with a steady and a pulsating inflow contains 97 % of the energy content.
- Coherent structures can be extracted by reconstructing the flow field with a truncated set of POD modes as shown for the ducted orifice plate.
- The energy content of the zero-th POD mode for the radial turbine is mainly located in the volute, while the energy of the first and second POD modes are located in the turbine wheel and represents the rotation of the wheel, implying that the rotation of the wheel is the main source of flow that deviates from its mean.

For the double bent pipe we have further found that:

- The non-linear terms of the Navier-Stokes equations have a negligible influence on the solution to the system of ODEs.
- The system of ODEs is inherently unstable and gets more unstable the higher the number of modes that are used.
- Modelling errors, due to the use of a truncated set of POD modes, will be amplified as the system of ODEs is integrated in time, if no damping is introduced into the system.

The approach is appropriate for non-trivial flows with strong coherent structures where just a few modes can capture the main energy content of the flow field, which is not the case for the pipe geometry under slow pulsating conditions. In contrast to tabulated data, the reduced model does account for flow unsteadiness in a natural manner. Since it is possible to reconstruct the
flow field in time from the computed time coefficients and the POD modes, the power output in the case of a turbine, can directly be computed. A disadvantage of the Galerkin-POD approach is the initial high computational cost to generate the POD modes.

The concept of using the Galerkin projection of the Navier-Stokes equations with POD modes as a basis have been shown for a double bent pipe chosen for the demonstration. Unfortunately, the results to predict the time coefficients of the POD have not been satisfying with respect to accuracy. The reason is that the flow in the double bent pipe does not have coherent structures that are strong enough to be captured by only a few modes. As a proposal for future work, the method should be used on a flow case with strong coherent structures, as the radial turbine due to the structures encountered in the wheel to show the potential of the method. It is expected that the approach would be beneficial for off-design conditions and situations when the flow inside the turbine separates in an unsteady manner.

The model using a few POD modes gives errors since a part of the flow field is neglected when a truncated number of POD modes are used. This could be overcome by integrating a few time steps of the LES into the time marching of the reduced model, i.e. the system of ODEs derived from the Galerkin projection, to improve the model’s accuracy. This approach could also be seen as a speed up of the LES with preserved accuracy. Another concern is the model’s operating range or geometrical modifications, since a single set of POD modes will most probably not cover the entire operational range or significant geometrical modifications. A model should therefore be built from several sets of operational points and geometries.
Acknowledgements

First of all, I would like to give my gratitudes to my always enthusiastic supervisor, Prof. Hans-Erik Ångström for guiding me throughout the wonderful world of engines and who made this project possible. To my encouraging advisor, Prof. Laszlo Fuchs for introducing me to the challenging and interesting field of LES and its possibilities. Your never ending flow of ideas have inspired me to be more curious than ever!

The Swedish Energy Agency (STEM) is acknowledged for the financial support. High Performance Computing Center North (HPC2N) and The National Supercomputer Centre in Sweden (NSC) are acknowledged for supplying all the necessary computer resources.

I would also like to thank all my colleagues at the div. of Internal Combustion Engines and at the dept. of Mechanics for making my days at school so joyful. I’ve enjoyed every minute of discussing physical phenomena that we encounter in our daily life to passenger car engines. A special thanks goes to Emma Alenius for all the discussions we’ve had, your indispensable help to find answers to all my queries and for giving me the data on the ducted orifice plate and to Fredrik Hellström for inviting me to our room, all the valuable help you’ve giving me at the start of the second part of the project and discussing IC-engines from a fluid mechanics perspective, I miss those days! And for giving me the turbine data. To my eldest friend, Jonas Bylund for helping me out any time I got into programming trouble (quite frequently) and for learning me the basics of object oriented programming. To Alexander Eiler for all the thousands of kilometres of trails we’ve biked together all over Uppland and at the more exotic places we’ve visited, it helped to clear my mind! (and for letting me win every now and then when going uphill)

Thanks to all my friends and family (incl. both brothers) for always being there for me, that you’re still being you and letting me being me! Last but not least, I would like to thank min Karolin for all the support and love you’ve giving me throughout the years, I wouldn’t have made it without you. P o g, D!

/Niki
References


CAPOBIANCO M. & GAMBAROTTA A. 1993 Performance of a twin-entry
REFERENCES

automotive turbocharger turbine ASME Energy-Sources Technology Conference 
& Exhibition, Paper No. 93-ICE-2

Orthogonal decomposition and low-dimensional models for driven cavity flows 

Chen H., Hakeem I. & Martinez-Botas R. F. 1996 Modelling of a turbocharger 
turbine under pulsating inlet conditions, Proceedings of the Institution of 
Mechanical Engineers, Part A: Journal of Power and Energy, Vol. 210, p. 397- 
408

Chen H. & Winterbone D. E. 1990 A method to predict performance of vaneless 
radial turbines under steady and unsteady flow conditions, Proceedings of the 
IMechE 4th International Conference on Turbocharging and Turbochargers Paper 
no. C405/008, p. 13-22

Costall A. 2007 A one-dimensional study of unsteady wave propagation in 
turbocharger turbines, Doctoral Thesis Imperial College, London

Pulse performance modeling of a twin-entry turbocharger turbine under full and 
unequal admission, Proceedings of ASME Turbo Expo 2009: Power for Land, 
Sea and Air, GT2009-59406

Dale A. & Watson N. 1986 Vaneless radial turbocharger turbine performance, 
IMechE Paper no. C110/86

Dale A.P. 1990 Radial, Vaneless Turbocharger Turbine Performance, Doctoral 
Thesis, Imperial College, London

EASY! For Turbines Tutorial April 2005 version 7.7.70 Concepts NREC

Ehrlich D.A., Lawless P.B. & Fleeter S. 1997 On-engine turbine inlet flow 
characterisation, SAE paper 971565


Hellström F. & Fuchs L. 2007 Numerical computations of steady and unsteady 
flow in bended pipes AIAA 2007-4350, 37th Fluid Dynamics Conference and 
Exhibit, Miami Fl. 2007

Hellström F. & Laszlo F. 2008 Effects of inlet conditions on the turbine 
performance of a radial turbine, In Proceedings of ASME Turbo expo 2008: 
Power for Land, Sea and Air, Berlin, GT2008-51088

Hellström F. & Fuchs L. 2008 Numerical computations of pulsatile flow in a turbo-
charger Proceedings of 46th Aerospace Sciences Meeting, January 2008, Reno, 
Nevada, AIAA 2008-735

Hellström F. & Fuchs L. 2009 Numerical computation of the pulsatile flow in a 
turbo-charger with realistic inflow conditions from an exhaust manifold ASME 
Turbo Expo 2009: Power for Land, Sea and Air June 8-12, 2009, Orlando, 
Florida

Hirsch C. 1984 Numerical computation of internal and external flows, volume 2: 
Computational methods for inviscid and viscous flowsby John Wiley & Sons Ltd

Holmes P., Lumley J. L. & Berkooz G 1996 Turbulence, Coherent structures, 
dynamical systems and symmetry Cambridge University Press, Cambridge 1996
REFERENCES


King A. J. 2002 A turbocharger unsteady performance model for the GT-Power internal combustion engine simulation, Doctoral Thesis, Purdue University


Laguna-Gómez Jose P. EU Legislative Perspective 2007 Heavy-Duty Emissions Control Symposium, Gothenburg 2007


Langridge S. and Fessler H. 2002 Strategies for high EGR rates in a Diesel Engine SAE paper no. 2002-01-0961

Lawless P.B. 1997 Characterization and modeling of turbocharger dynamic performance, SAE paper 971566


Macek J. & Vítek O. 2008 Simulation of pulsating flow unsteady operation of a turbocharger radial turbine, SAE paper 2008-01-0295


REFERENCES

Muld T. W. 2010 Analysis of flow structures in wake flows for train aerodynamics Licentiate thesis in Mechanics, Royal Institute of Technology, Stockholm 2010
Pope S. B. 2000 Turbulent flows Cambridge University Press
Renberg U., Ångström H-E. & Fuchs L. 2009 Comparative study between 1D and 3D computational results for turbulent flow in an exhaust manifold and in bent pipes SAE Paper no. 2009-01-1112
Rowley C. W., Mezic I., Bagheri S., Schlatter P. & Henningson D. S. 2009 Spectral analysis of nonlinear flows JFM under consideration
REFERENCES


WESTIN F. 2005 Simulation of turbocharged SI-engines - with focus on the turbine Doctoral Thesis, Royal Institute of Technology - KTH, Stockholm


