Energy Conversion and Management 86 (2014) 520–536

Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

# Flow effects due to pulsation in an internal combustion engine exhaust port

Bernhard Semlitsch[a,*], Yue Wang[b], Mihai Mihăescu[a]

[a] Linné FLOW Center – Competence Center for Gas Exchange (CCGEx), Department of Mechanics, Royal Institute of Technology (KTH), Osquars Backe 18, Stockholm 10044, Sweden
[b] National Key Laboratory of Aerodynamic Design and Research, Northwestern Polytechnical University, Youyi West Road, No. 127, Xi'an, Shaanxi 710072, China

ARTICLE INFO

*Article history:*
Received 25 April 2014
Accepted 11 June 2014
Available online 27 June 2014

*Keywords:*
Internal combustion engines
Fuel economy
Turbocharged engines
Exhaust gas energy
Automotive exhaust systems

ABSTRACT

In an internal combustion engine, the residual energy remaining after combustion in the exhaust gasses can be partially recovered by a downstream arranged device. The exhaust port represents the passage guiding the exhaust gasses from the combustion chamber to the energy recovering device, e.g. a turbocharger. Thus, energy losses in the course of transmission shall be reduced as much as possible. However, in one-dimensional engine models used for engine design, the exhaust port is reduced to its discharge coefficient, which is commonly measured under constant inflow conditions neglecting engine-like flow pulsation. In this present study, the influence of different boundary conditions on the energy losses and flow development during the exhaust stroke are analyzed numerically regarding two cases, i.e. using simple constant and pulsating boundary conditions. The compressible flow in an exhaust port geometry of a truck engine is investigated using three-dimensional Large Eddy Simulations (LES). The results contrast the importance of applying engine-like boundary conditions in order to estimate accurately the flow induced losses and the discharge coefficient of the exhaust port. The instantaneous flow field alters significantly when pulsating boundary conditions are applied. Thus, the induced losses by the unsteady flow motion and the secondary flow motion are increased with inflow pulsations. The discharge coefficient decreased about 2% with flow pulsation. A modal flow decomposition method, i.e. Proper Orthogonal Decomposition (POD), is used to analyze the coherent structures induced with the particular inflow and outflow conditions. The differences in the flow field for different boundary conditions suggest to incorporate a modeling parameter accounting for the quality of the flow at the turbocharger turbine inlet in one-dimensional simulations.

## 1. Introduction

Approximately 20–40% of the total provided energy after combustion is lost in the exhaust gasses [1,2]. However, the residual energy content of the exhaust gasses can be split into two parts, thermal energy and pressure energy. The exhaust gasses in the cylinder after combustion are hot and the gasses contain therefore potential usable energy, which can be extracted in a further process [3]. Thus, turbocharger are often used to expand in part the heat and the pressure potential [4], in order to increase the specific engine efficiency.

The exhaust stroke of a four stroke engine can be divided into two main periods, the blow-down and the scavenging pulse. The pressure difference between the in-cylinder pressure and the pressure in the exhaust tract forces the exhaust gas expulsion at the early stage, which is referred as the blow-down phase. When the valves start to open, only a small slid between the valve head and the valve seat represents the discharge passage. Nevertheless, during the blow-down phase, the mass flow rate is high and decreases rapidly with the pressure dropping in the cylinder pod. Thereafter, when the pressure difference between the cylinder chamber and the exhaust manifold is balanced, the scavenging phase takes place, in which the piston cleans out the rest of the residual gas by its motion towards the top dead center. During this phase, the mass flow rate is lower than during the blow-down phase. However, the response of the mass flow rate during the scavenging phase depends highly on the pressure conditions downstream in the collecting manifold, the possible interactions with other cylinders over the manifold, and the back-pressure provoked by the turbocharger.

The turbine of the turbocharger extracts flow energy and generates thereby a resistance to the flow downstream of the exhaust port. The flow resistance induces a back-pressure, which complicates the

---

* Corresponding author. Tel.: +46 8 7907157; fax: +46 8 7969850.
*E-mail address:* bernhard@mech.kth.se (B. Semlitsch).

http://dx.doi.org/10.1016/j.enconman.2014.06.034
0196-8904/© 2014 Elsevier Ltd. All rights reserved.

gas expulsion from the cylinder since this results in additional work demanded from the piston. In order to reduce the back-pressure effect of the turbocharger, several approaches can be employed. A waste gate to bypass the turbocharger during the scavenging phase can be used to reduce the back-pressure. With variable valve actuation and divided exhaust periods two separated exhaust ports are used for the blow-down phase and the scavenging phase, where the valves are actuated at different times [5]. Variable valve timing, i.e. regulating adaptively the valve opening time, the opening duration, and the valve lift, is crucial for the engine performance [6] and can be used to reduce the fuel consumption of the internal combustion engine [7]. However, the alteration of the valve timing influences the emission production. Simulating the entire process of an internal combustion engine with all degrees of freedom is computationally unaffordable. Therefore, fast simplified models are utilized for the initial design process of the internal combustion engine [8]. However, the physics and the behavior of the engine must be represented by the model. An oversimplified model may not represent the real engine any more. The inflow conditions into the turbocharger turbine can crucially effect its performance [9]. Usually, the valve timing and the valve opening-speed is studied and optimized by one-dimensional simulations, where the quality of the flow field and therefore the inflow conditions into the turbocharger turbine are not accounted for.

In one-dimensional modeling of the internal combustion engine process, tables, databases, and operation maps are used to describe the performance behavior of the individual constituents, such as the piping system, constrictions, or bends [10,11]. The tables or databases are based on empirical formulas with experimentally evaluated coefficients. For the exhaust port, the discharge coefficient is evaluated and employed in the one-dimensional engine analysis to evaluate the total pressure drop [12]. A so-called flow bench experiment is performed, where the exhaust port discharge coefficient is evaluated at fixed valve lifts and constant specified total pressure drops. These experiments are usually performed on the real cylinder head at room temperatures. Hence, the flow streaming through the port is cold in contrast to a real engine case [13]. The discharge coefficient of sharp edged flow constrictions was evaluated experimentally [14], with the conclusion that the steady discharge coefficients are generally lower than those under unsteady flow conditions. The effect of pulsating flow on the exhaust port flow coefficients has been investigated numerically for different geometries regarding the blow down pulse and room temperatures [15]. It was found that the maximum difference in the flow coefficient was at most a 6% increase or a 7% decrease. The averaged flow coefficient over several valve lifts varied between 0.5% and 2.5% for the different geometries, which resembles a low impact on the total engine performance. Therefore, Bohac and Landfahrer [15] conclude that flow bench measurements at constant valve lift with constant mass flow rate are adequate.

Continuous ordinary differential equations can be used in one-dimensional simulation codes for more accurate modeling of engine components [16] or power plants [17]. The transfer functions for a given geometry can be obtained by impulse excitation to determine the characteristics of an engine transmission component. An experimental approach relating the mass flow rate with the dynamic pressure fluctuations can be used to abstract an ordinary differential equation model for an engine part [18]. A flow based model can be obtained reducing the Navier–Stokes equations via Galerkin projection onto representative flow modes [19]. Thus, the original governing system of partial differential equations is replaced by a set of ordinary differential equations, which are computationally inexpensive to compute. The Proper Orthogonal Decomposition (POD) method decomposes the flow field into a series of modes, where the modes are constructed such that the least number of modes is needed to reproduce the kinetic energy of the flow field [20]. The POD modes are also used to characterize the flow [21] and identify coherent structures with large energy contend. Also Fourier mode decomposition or dynamic mode decomposition have been used for constructing reduced order models [22], where certain frequencies are important [23].

The experimental assessment of the flow field in a complex, confined geometry, such as the exhaust port, is a challenging task [24]. However, experimental flow visualizations of the flow field development inside complex geometries, such as a closed cylinder with moving piston and combustion, have been performed [25]. Further, experimental flow visualization in an exhaust port can be performed with a considerable effort and assumptions [26], such as fixed valve lifts. Nevertheless, numerical computations of the flow in a complex geometry with non-moving boundaries can be easily performed. The computational effort of a numerical simulation depends on the complexity, amount of modeling and flow scales considered. A time-averaged simulation approach, such as steady state Reynolds Averaged Navier–Stokes (RANS) simulations, can be achieved with a rather low computational afford. However, the required time resolution of the flow phenomena for POD is not computed. To resolve all flow scales, i.e. Direct Numerical Simulation (DNS), is computationally expensive and the present purpose does not justify this the excessive usage of resources. With a reasonably resolved LES simulation, a substantial proportion of the inertial subrange in the turbulence spectra is resolved and only the smallest flow scales are modeled. The smallest flow scales dissipate the flow energy to heat and this behavior is of universal character, independent of the geometry. Generally, large energetic turbulent structures imply a substantial proportion of turbulent dissipation [27]. Hence, the LES approach reassembles a suitable trade between accuracy, reliability, and effort, which has been successfully applied to optimize the port geometry of internal combustion engines [28]. Many numerical investigations analyze the flow in the intake port and the consequences on the in-cylinder flow. Additionally, the flow in the ports of an internal combustion engine with pulsating boundary conditions has been simulated, e.g. [28]. However, the most studies emphasize on the computation models, performance parameter, or optimization of the geometry rather than giving insight into the generated flow structures occurring in the exhaust port.

This analysis is part of a larger investigation study treating the gas exchange process in an internal combustion engine. Within this paper the importance of dynamic boundary conditions on the flow generated losses in a realistic exhaust port of an internal combustion engine is investigated numerically. The three-dimensional compressible Navier–Stokes equations are simulated using an LES approach. A comparison of two cases, i.e. using constant inflow and outflow boundary conditions, and using engine cycle dependent inflow boundary conditions, at a constant valve lift are performed. These assumptions allow to isolate the effects caused due to flow pulsation. Therefore, the focus of contrasting the cases is held on the quantities, which are characteristic for the flow induced losses, as e.g. turbulent dissipation and friction losses. The change of the coherent flow structures are analyzed using the POD method.

## 2. Case description

The exhaust port geometry studied in this numerical investigation stems from a real internal combustion engine, the SCANIA D12, including the cylinder and the exhaust port valves. The relevant specifications of the internal combustion engine are tabulated in Table 1. The geometry used for the computational simulations is depicted in Fig. 1. On top of the engine cylinder, two exhaust ports are situated off-center, each in a quarter section of the circular