<http://www.diva-portal.org>

Postprint

This is the accepted version of a paper presented at *4th International Conference on Jets, Wakes and Separated Flows, ICJWSF2013, September 17-21, 2013, Nagoya, Japan..*

Citation for the original published paper:

Wang, Y., Semlitsch, B., Mihaescu, M., Fuchs, L. (2013)
Flow-structures Generated by Valve and Piston Motion in an Exhaust Port of a Truck Engine.
In: *Proceedings of the 4th International Conference on Jets, Wakes and Separated Flows-IV*

N.B. When citing this work, cite the original published paper.

Permanent link to this version:

<http://urn.kb.se/resolve?urn=urn:nbn:se:kth:diva-134840>

FLOW-STRUCTURES GENERATED BY VALVE AND PISTON MOTION IN AN EXHAUST PORT OF A TRUCK ENGINE

^{*1}Yue Wang, ^{*2}Bernhard Semlitsch, ^{*1,2}Mihai Mihaescu, ^{*1,2}Laszlo Fuchs

^{*1}CCGEx, Department of Mechanics, KTH Royal Institute of Technology
Osquars Backe 18, Stockholm SE-100 44, Sweden
yue@mech.kth.se

^{*2}Linné Flow Center, Department of Mechanics, KTH Royal Institute of Technology
Osquars Backe 18, Stockholm SE-100 44, Sweden

ABSTRACT

The exhaust port of a truck internal combustion engine forms the interface between the combustion engine and the turbocharger. Approximately 30-40% of the energy potential is lost in the exhaust gasses after combustion, which can be partially recuperated in a turbocharger. Hence, energy losses in the connection are highly undesired. However, due to the high occurring velocities and the complex geometry, flow separation, flow-structure formation, and secondary flow motion are the major sources of energy losses.

Within the exhaust process, the valves open while the piston continues moving in the combustion chamber. This process is often analyzed modeling the piston and valves at fixed locations, but conserving the total mass flow. Using advanced methods, this process can be simulated numerically in a more accurate manner.

This study compares Large Eddy Simulations assessing the implied differences due to the methodical of simulating the exhaust process from an engine cylinder. A simple case using fixed positions for valve and piston is contrasted with the cases of static valve and moving piston, and moving valve and moving piston. The generated flow phenomena are compared within the cases.

1. INTRODUCTION

In the last years, the focus of internal combustion engine development has moved towards more efficient engines and at the same time reducing the environmental impact caused by the pollutants, where their source is the combustion. From the provided energy by combustion in a diesel engine, approximately 30-40% of the energy is extinct in the burned gases [1]. However, the gases are hot and therefore rich of energy, which can be utilized. To increase engine efficiency the hot exhaust gases can be recycled in a turbocharger.

During the outtake stroke the used gas is expelled from the cylinder. This stroke consists of mainly two periods, the blow-down and the scavenging [2]. In the blow-down phase, the pressure differential between the

combustion chamber and the exhaust port drives the expulsion of the used gas. Thereafter, follows the scavenging, where the piston cleans out the rest of the gas by its motion. In the initial blow-down phase the valves start to open. Hence, the cross-sectional area is small, but the pressure difference causes high velocities. Thus, this phase is most delicate to provoke choked flow, which results in high pressure losses and therefore, a high discharge coefficient for the exhaust pipe.

In one dimensional internal combustion engine analysis utilize databases and operation maps for the individual components, as e.g. bent pipes, compressors or turbines. The exhaust port including the valves is too complex to be approximated from a database. Energy is lost in the exhaust port, either directly or through formation of flow structures that enhance dissipation. Furthermore, it has been shown by several researchers that the efficiency of the turbocharger is dependent on the flow structures at the turbine inlet [3, 4]. Commonly, the exhaust port is modeled by experimentally evaluated discharge coefficients, which are obtained by measurements for fixed valve lifts at for a low pressure-drop. Using these values for an arbitrary valve motion the total pressure loss is approximated. However, higher pressure-drops may occur when the flow chokes. Hence, this procedure might lead to remarkable errors, when the flow bursts through the port at small valve lifts.

However, a drawback of the turbocharger is that it increases the back pressure in the exhaust port. This has a negative effect on the gas expulsion from the cylinder. There are several strategies to overcome this drawback, as e.g. using a waste gate to decrease the back pressure in the last part of the stroke, variable valve actuation and divided exhaust periods, where two separated exhaust ports for blow-down and scavenging are used and the valves open at different times. However, the motion curves describing the lift of the valves are critical for the generated pressure loss and thus, for the efficiency of the internal combustion engine. Since, the valve timing and the valve opening-speed is studied and optimized mostly by one-dimensional simulations, a dynamic discharge co-

Table 1 Specifications of the internal combustion engine

Engine Label	Scania D12
Displacement (dm^3)	11.7
Bore (mm)	127
Stroke (mm)	154
Conn. Rod length (mm)	255
Compression ration	18
EVO (CAD)	136
EVC (CAD)	359

efficient would be required to obtain most accurate results.

Accurately predicting heat transfer from the hot cylinder to the exhaust port is a significant problem in engine design. The temperature difference is an important parameter for the assessment of the performance of the turbine and EGR cooler. The heat transfer in simple geometries has been studied and tabulated for engineering purposes, where correction parameters for more complex geometries can be used. For complicated flows like valve ports, this approach tends to fail in predicting the heat flux correctly [5]. The contributions to heat transfer from large-scale fluid motion indeed dominates over the near wall boundary layer [5].

The purpose of this paper is to characterize the impact on the flow and the losses in an exhaust port, which is generated by the dynamic valve motion and piston motion. Better prediction of the dynamic losses is required to develop more efficient internal combustion engines, which is assessed in this paper. The compressible Navier-Stokes equations have been simulated by a finite volume code using the Large Eddy Simulation approach to handle turbulence. Four cases have been simulated regarding valve and piston motion during the outtake process in a realistic geometry of the internal combustion engine. The valve and piston motion is modeled by different numerical modeling approaches and the effect of the valve and piston motion is analyzed.

2. GEOMETRY AND CASES

The exhaust port of an internal combustion heavy-duty Diesel engine, the D12 from Scania, has been the object of the numerical study. The production of flow losses while the used gas exhausts the cylinder is analyzed only for one exhaust port, since the losses generated in the other cylinders should behave similar. The geometry consists of five parts, a cylinder, and a complex port with two pipes conjunct at one end, two valves and a long exit pipe as shown in Fig.1. The fictitious exit pipe was attached at the end of the exhaust port and has a length of the exhaust pipe diameters. The detailed parameters of the internal combustion engine are specified in Tab.1 .

Three different simulation approaches have been investigated, where an overview of the simulated cases can

Fig. 1 The geometry of the internal combustion engine is shown. A: The front view; B: The top view; C: The cross section of valve 3. 1: Cylinder; 2 and 3: valves; 4: Exhaust port; 5: exit pipe.

be found in Tab.2. In all simulated cases, the walls (including the valves) are assumed to ideally smooth and a no-slip condition was applied. The values for the boundary conditions used here are taken from one-dimensional simulations. Due to external cooling of certain parts, the wall temperature conditions are fixed at different temperatures: the piston at 600 K, the cylinder wall at 400 K, and the port wall at 500 K. The valves are assumed to behave adiabatic. The outlet boundary condition is modeled by a static temperature 600K and a static pressure outlet of 200 kPa, resembling the back pressure of the turbocharger.

The simplest modeling approach is represented by the cases ignoring the motion of the valve and the piston. Hence, the valve is fixed at a certain valve lift position of 5 mm and the piston is resembled by a total pressure source acting in the cylinder. The total pressure source is held constant during several flow cycles. Hence, initial blow down effects are not covered within these simulations. During the engine cycle, two times a valve lift of 5 mm is reached, one time in the opening phase and another time during the closing phase. The pressure difference is different for these two cases. Therefore, two cases have been simulated resembling the two conditions. The initial values for the total pressure have taken from the moving valve moving piston case at this valve lift, which is described later on. This simulation resembles the case evaluated experimentally to measure the discharge coefficient for a certain valve lift, where dynamic flow features of valve and piston motion are ignored.

In a further step, the piston motion has been simulated within case C2. The piston follows the motion displayed in Fig.2. The valve starts at an opening lift of 3.5 mm and stops its motion at 5 mm valve lift. Only the outtake of the used gas is simulated. Hence, the piston moves from the bottom dead center point to the top dead center point. The flow-field and all flow quantities are kept for the next cycle, whereas the cylinder is refilled with the initial conditions, where this approach is sketched in Fig.3. The initial conditions in the cylinder are a static pressure of 390 kPa and temperature of 820

Table 2 Overview of simulated cases

Cases	Piston	Valve
C1	Moving	Moving
C2	Moving	Fixed at 5mm
C3	Total pressure 340 kPa	Fixed at 5mm
C4	Total pressure 220 kPa	Fixed at 5mm

K for each simulated cycle.

The third and most advanced approach used for case C1, includes the full valve and piston motion. The valve and piston motion is handled as visualized in Fig.2. The same boundary conditions as in the previous described case. The valves never close entirely. To be able to handle the complete valve closing in numerical simulations advanced requirements are requested from the computational method, which is computationally very expensive.

Fig. 2 Piston and valve lift are shown as a function of crank angle: illustrated in black the piston motion (cm); red displayed the valve lift for C1; red dash line corresponds to the fixed valve lift at 5mm for case 2.

3. NUMERICS AND GRIDS

All numerical simulations were preformed using a commercial compressible finite volume code starccm+ [6]. The cases described in the previous section were investigated using Large Eddy Simulations (LES). For time advancement a second order implicit scheme was used with explicit time-stepping of $\Delta t = 10^{-7}s$ and a bounded second order central-difference scheme for spatial discretization was applied [6].

3.1 Turbulence modeling

The turbulent scales generated in the geometry are quite small. The boundary layers and the flow-scales close to the walls are very small compared to the domain extensions. To capture all flow features occurring

Fig. 3 Illustration of the Initial conditions for each simulation cycle. A: Static pressure distribution at 324 CAD the end of the last simulation. B: Static pressure distribution at 171 CAD for the following simulation cycle.

a very fine mesh would be required. Such fine computational grids lead to a high numerical expense. To achieve qualitative good results in a reasonable amount of resources and time only the flow characteristic turbulent scales have been simulated. About an order of magnitude of the internal sub range has been resolved with the used numerical grid. Such an approach is commonly known a LES, where only the important scales are considered and the smaller scales are neglected. Due to such a discretization of the governing equations an additional term appears, the sub grid scale term. Using different assumptions regarding this term, models can be applied. However, using the general assumptions for this term seems a very crude approximation, since in such a complex geometry and flow the most simplifications do not apply. Hence, no explicit expression for the sub grid scale term is employed. The employed approach uses the inherent numerical dissipation of the numerical scheme to account for the effect of the unresolved sub grid dissipation. Such an LES is commonly known as implicit LES (ILES). Many researchers have shown that a second order accurate scheme is sufficient to be considered suitable for the ILES approach [7, 8, 9]. Implicit LES has been successfully applied for vast range of fluid dynamic problems using a variety of different solvers [10, 11].

3.2 Computational grids

The numerical meshes used in the simulations have a hexahedral, isotropic core-mesh and two prism layers next to the walls. The hexahedral cell-core size is about 0.6 mm. The two prism layers have the total thickness of 0.3 mm and the stretching ratio of them is 1.2. Additional mesh stretching is applied to the final section of the outlet domain to damp reflections at the outlet. In total, the computational grid consists of six million cells.

4. RESULTS

In this section the results obtained by the compressible LES are presented.

The initial static pressure in the cylinder is updated to 3.9 bar at each engine cycle at a static temperature of 820 K. Figure 4 illustrates the phase-averaged total pressure in the cylinder and at a cross-section close to the outlet as a function of the crank angle. The total pressure in the cylinder decreases while the flow bursts out of the cylinder. The pressure wave propagating through the exhaust port reaches with some delay the outlet. A high peak response when the pressure wave hits the monitoring cross-section can be seen in Fig.4.

Four crank angles have been selected for the flow-field comparison, 178°, 216°, 261° and 317°, as indicated with dash lines in Fig.4. The boundary conditions for the cases C1 and C2 are the same until a crank angle of 178°, hence, there are no difference observable for these cases until this point. Figure 5 illustrates Mach-number distribution in the flow domain. The flow bursts out of the cylinder, and is confronted with the contraction generated by the shape of the cylinder and the valve edge. Due to the high pressure difference and the contraction, the phase averaged velocity magnitude reaches up to 300 m/s in the gap for the moving valve and moving piston case. However, the Mach-number stays below 0.7 for these cases. Modeling the valve fixed and the piston as a mass-flow inlet, the Mach-number rises to values slightly higher than one. Thus, the flow is choked, which can be clearly seen in Fig.5. Due to the choked flow, the temperature downstream differs highly between the cases. Flow separation and turbulent properties exhibit a complete different behavior.

The seat and the valve edge shape the gap contour. The high velocity flow hits the valve seat at the cut, which generates two high wall shear stress regions, which is illustrated in Fig.6. The collision causes the flow to change direction and generates flow separation with flow reversal. After the flow passed the smallest cross-section, created by the valve and the port, the flow expands into the exhaust port. The flow is decelerated due to the increase in the cross-sectional area. Due to the presence of the valve, an annular, high velocity, unsteady, jet-like flow structure is formed above the top surface of the valve. The flow collides with the valve stem and is bended along the valve stem. The flow reaches the top of the valve stem with a curvature along the outer bend of the port. The flow coming from the other valve port channel exhibits similar flow features. The flows from two channels are collected and contracted into the exit pipe.

An interesting comparison between case C1 (moving valve and moving piston) and C3 (stagnation pressure inlet) is to access the difference caused by the piston and valve motion. The total pressure in the cylinder is 340 kPa for both cases, as shown in Fig.8. However, in the exhaust port and exit pipe, the total pressure distribution is quite different from these cases. Obviously the

Fig. 4 Phase averaged total pressure at measured in the cylinder and at a cross-section before the outlet. Black curves: C1, moving valve moving piston; Blue curves: C2, fixed valve at 5mm and moving piston.

Fig. 5 Mach-number distribution (m/s) at a crank angle of 178° corresponding to a valve lift of 5mm for case C1, C2 and C3.

Fig. 6 Showing the instantaneous wall shear stress at a crank angle of 178° for the cases C1 and C2 (Pa).

total pressure in the port for C1 is higher than for case C3.

Piston moves up after a crank angle of 180° and the blow-down pulse ends at a crank angle of about 220° (see

Fig. 7 Phase averaged pressure distribution (Pa).

Fig. 8 Comparison of total pressure distributions (Pa) between the cases, phase averaged total pressure at 178° for case C1 (A) and time-averaged total pressure distribution for case C3 (B).

Fig.4). For case C1, at a crank angle of 216° , the total pressure in the cylinder is lower than the total pressure in the port and exit pipe (Fig.4). Figure 9A shows the phase averaged velocity magnitude distribution for the case C1. The valve lift is increased to 12 mm. The maximum velocity magnitudes decrease to 100 m/s. In the gap, the recirculation bubble due the contraction mechanism is clearly shown on the top surface of the valve in Fig.9. The low velocity region due to the cut of the valve seat still exists there in cut-plane 1. The negative velocity region in the port shrinks and nearly disappears. The different behavior is found for case C2. A large negative velocity region is found in the plot of the cross-sections 2 and 3, which are shown in Fig.9B.

At the crank angle of 247.5° , the valves reach the maximum valve lift and return to closing position. Figure 10A shows the velocity distribution at crank angle of 261° for case C1. The compression mechanism of the piston increases the pressure in the cylinder and the maximum velocity of 150 m/s. Figure 10B shows the velocity distribution in case C2. From the comparison of Fig.10A and Fig.10B, the low velocity region in the port can be found in Fig.10B, but not in Fig.10A. And the velocity in the exit pipe of case C1 is larger than the velocity of case C2.

Finally, at a crank angle of 317° , the valve lift has reached again the value of 5 mm. In case of moving valve and moving piston the velocity distribution shows similar shape than for the case with piston motion but without valve motion. However, the amplitudes of the

Fig. 9 Phase-averaged velocity magnitude distribution (m/s) across a cross-section through the valve at a crank angle of 220° . Subplots 1 to 6 show stream-wise velocity distribution. A: Case C1 ; B: Case C2.

velocities behave rather different. The comparison of the phase-averaged velocity distribution is shown in Fig.11.

5. CONCLUSION

The losses caused by the dynamics of the valve while the exhausting gas passes, are often neglected. The relevance of modeling valve and piston motion during the outtake of the used gasses in an exhaust port of a heavy-duty internal combustion engine has been studied using LES simulations. Three different approaches have been used to model this process. A comparison between the approaches of modeling the valve at a fixed location using a mass-flow to resemble the piston, simulating the piston motion and keeping the valve fixed, and simulate the moving valve and the motion of piston have been performed.

It has been found that the flow-field is predicted at two high velocities by modeling the valve at a constant lift. For the static case (fixed valve and modeled piston) the flow chokes, while the Mach-number stays for the moving valve and moving piston in a regime below 0.6 at the same valve lift. The distribution of total pressure for these two cases shows a similar shape, but the amplitudes are distinct. Separation bubbles form more clearly in the case of fixed valve fixed piston than for the moving case.

Fig. 10 Phase-averaged velocity magnitude distribution (m/s) across the valve 3 at 261° CA; plots 1 to 6 are stream-wise velocity distribution. A: Case C1 ; B: Case C2.

6. ACKNOWLEDGEMENT

This work was supported by the Competence Center for Gas Exchange (CCGEx) at KTH and Chinese Scholarship Council (CSC). The computational resources provided through the Swedish National Infrastructure for Computing (SNIC 002-12-11 & SNIC 2013/1-72) are greatly acknowledged.

REFERENCES

- [1] Baines C. N., Fundamentals of Turbocharging, Vermont: Concepts NREC, (2005).
- [2] Heywood John B., Internal Combustion Engine Fundamentals, McFraw-Hill, Inc, (1988).
- [3] Ehrlich D., Lawless P., Fleeter S., On-engine Turbocharger turbine inlet flow characterization, In 48th Earthmoving Industry Conference & Exposition, Society of Automotive Engineers, 400 Commonwealth Dr, Warrendale, PA, 15096, USA. (1997).
- [4] Hellström F., Fuchs L., Effects of inlet conditions on the performance of a radial turbine, In ASME Turbo Exposition, (2008).
- [5] Caton J. A., Heywood J. B., An experimental and analytical study of heat transfer in an engine exhaust

Fig. 11 Phase-averaged velocity magnitude distribution (m/s) across the valve 3 at 317° CA; plots 1 to 6 are stream-wise velocity distribution. A: Case C1 ; B: Case C2.

port, International Journal of Heat and Mass Transfer 24 (4), 581-595, (1981).

- [6] CD-Adapco, Starccm+ User Guide 6.06.017, Tech. Rep., (2011).
- [7] Wilcox D. C., Turbulence modeling for CFD, 3rd edition. DCW Industries, (2006).
- [8] Drikakis D., Advances in turbulent flow computations using high-resolution methods, Process in Aerospace Sciences 39, 405-424, (2003).
- [9] Harten A., High resolution schemes for hyperbolic conservation laws, Journal of Computational Physics 48, 357-393, (1983).
- [10] Margolin L. G., Rider W. J., Grinstein F. F., Modeling turbulent flow with implicit LES, Journal of Turbulence 7, 1-27, (2006).
- [11] Garmann D., Visbal, M. and Orkwis P. D., Comparative Study of implicit and Subgrid-Scale Model LES Techniques for Low-Reynolds Number Airfoil Applications, International journal for numerical methods in fluids. (2012).