

CFD-Based Optimization of a Diesel-fueled Free Piston Engine Prototype for Conventional and HCCI Combustion

Miriam Bergman, Jakob Fredriksson and Valeri I. Golovitchev
Chalmers University of Technology

Copyright © 2008 SAE International

ABSTRACT

This paper presents results of a parametric CFD modeling study of a prototype Free Piston Engine (FPE), designed for application in a series hybrid electric vehicle. Since the piston motion is governed by Newton's second law, accounting for the forces acting on the piston/translator, i.e. friction forces, electrical forces, and in-cylinder gas forces, having a high-level control system is vital. The control system changes the electrical force applied during the stroke, thus obtaining the desired compression ratio. Identical control algorithms were implemented in a MATLAB/SIMULINK model to those applied in the prototype engine. The ignition delay and heat release data used in the MATLAB/SIMULINK model are predicted by the KIVA-3V CFD code which incorporates detailed chemical kinetics (305 reactions among 70 species).

Since the piston motion and frequency, the rate of heat release and the initial in-cylinder conditions all affect each other, while predicted using different modelling tools with no direct coupling between them, an iterative procedure was used among models describing:

1. Piston dynamics governed by Newton's second law including a high-level control system (using MATLAB/SIMULINK)
2. Combustion processes (using KIVA-3V)
3. Intake and exhaust system dynamics (using the GT-POWER module of the GT-SUITE™)

Effects of varying parameters such as compression ratios, power supplied to the compressor, fuel injection timings and injection pressures have been studied in both conventional diesel and HCCI modes, the target being to identify optimal conditions for the combustion process in which the engine can be operated highly efficiently with very low-emissions.

INTRODUCTION

THE FREE PISTON ENGINE PROTOTYPE

Interest has significantly increased recently in developing devices based on the Free Piston Engine (FPE) concept, designed to generate electricity in series hybrid electric vehicles [1,2,3,4], i.e. linear, crankless, internal-combustion engines. The first laboratory test for the FPE prototype installed at Chalmers University were conducted in conventional diesel mode at low load, in 2007.

The prototype was designed during the course of the European Commission-funded Free Piston Energy Converter (FPEC) project, part of the broader EU growth project G3RD-CT-2002-00814. Several previous combustion modeling studies for this device have been presented, e.g. [5,6,7]. The engine features two pistons connected by a shaft with permanent magnets fixed to it. Combustion occurs alternately at each cylinder in a two-stroke cycle, driving the magnets back and forth through the coils of an alternator. The alternator is used for both starting up the device and for controlling the motion of the translator. The layout of the FPE is schematically illustrated in Figure 1.

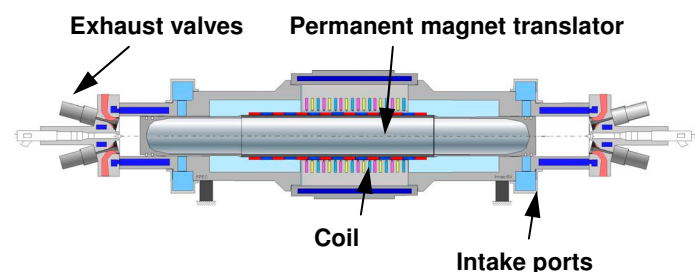


Figure 1. Schematic layout of the FPE

The FPE runs on a two-stroke cycle, employing the uniflow scavenging mode, using intake ports and exhaust valves. Direct fuel injection avoids short-circuiting of the air/fuel mixture, and allows for both conventional diesel and Homogeneous Charged Compression Ignition (HCCI) modes of operation.

Conventional diesel combustion regimes are feasible, but the HCCI mode is more desirable from an emissions perspective. One severe drawback with HCCI combustion is that it is difficult to control the heat release phasing, but the variable compression ratio operation of the FPE can provide means to overcome this difficulty, providing opportunities for flexible auto-ignition control on a cycle-to-cycle basis, without any hardware modifications. The ultimate goal is to achieve indicated fuel conversion efficiencies exceeding 50%, together with negligible amounts of undesired emissions.

METHODOLOGY AND COMPUTATIONAL TOOLS

Since there is no direct coupling between the different modeling tools, an iterative procedure was used, as illustrated in Figure 2, iterating between models describing:

1. Piston dynamics

The free piston motion is governed by Newton's second law, calculated using a MATLAB/SIMULINK model. The model incorporates a high-level control system, which changes the electrical force applied during the stroke, thus obtaining the desired compression ratio (r_c). Identical control algorithms were implemented in the MATLAB/SIMULINK model to those applied in the prototype engine.

2. Intake and exhaust system dynamics

The initial in-cylinder pressure and temperature conditions are obtained using the GT-POWER code. The model uses one-dimensional (1D) CFD, capturing the gas dynamics of the intake and exhaust systems.

3. Combustion processes

The Rate Of Heat Release (ROHR) and Start Of Combustion (SOC) were predicted from CFD simulations using a modified version of the KIVA-3V, rel. 2 code [8] incorporating a detailed chemistry mechanism (305 reactions). The models used in the CFD simulations at Chalmers University have been described in detail by various authors, e.g. [9].

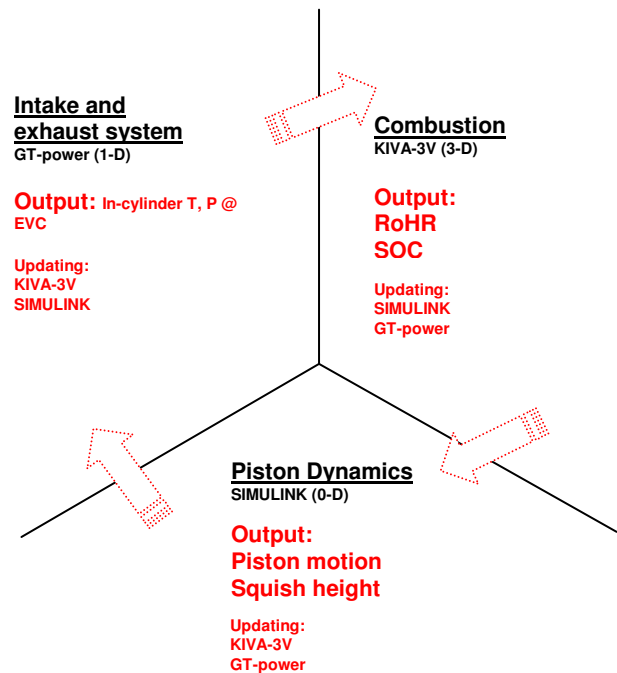


Figure 2. Methodology – schematic diagram of the iterative procedure

COMPUTATIONAL MODELS

DIESEL OIL SURROGATES

Diesel fuel is largely composed of higher hydrocarbons (C_{10} to C_{20}) with 50-65% alkanes, 20-30% cycloalkanes and 10-30% aromatics by volume [10]. Because of the complexity of real diesel oil composition and combustion kinetics, fuel surrogate models are used in advanced CFD simulations. Surrogate fuels consists of a small number of selected species that match real fuels in terms of spray atomization, droplet evaporation, ignition, combustion and emission formation processes. The current state-of-the-art in constructing and applying diesel oil surrogate models is summarized in [11].

The Diesel Oil Surrogate (DOS) model used in this paper uses the properties of Diesel fuel, given the formula $C_{14}H_{28}$. Four global reactions are used in the model, together with ~300 elementary reactions, including the detailed oxidation paths of n-heptane (C_7H_{16}) and toluene (C_7H_8), which were systematically reduced. Details of the mechanism construction can be found in [12]. The global reactions and their model constants are listed in Table 1.

Table 1. The global reactions (forward reaction rate: $k_f = A_f T^n \exp(-E_f/RT)$ in cm^3 , mole, sec cal units). A_{E_f} and A_{E_o} denote concentration exponents for fuel and oxidizer, respectively.

#	Global Reaction	A_f	n_f	E_f	A_{E_f}	A_{E_o}
1	$6\text{C}_{14}\text{H}_{28} + 7\text{O}_2 \Rightarrow \text{C}_7\text{H}_8 + 10\text{C}_7\text{H}_{16} + 7\text{CO}_2$	10^{13}	0	36500 (Convent. Comb.) 32000 (HCCI)	0.25	1.5
2	$4\text{C}_7\text{H}_{16} + 23\text{O}_2 \Rightarrow 28\text{HCO} + 18\text{H}_2\text{O}$	$5 \cdot 10^{11}$	0	28000	0.25	1.25
3	$\text{A2R5} \Rightarrow 12\text{C(S)} + 4\text{H}_2$	$2 \cdot 10^3$	0	0	1	-
4	$\text{C}_4\text{H}_2 \Rightarrow 4\text{C(S)} + \text{H}_2$	$2 \cdot 10^4$	0	0	1	-

The first global reaction above decomposes the starting fuel ($\text{C}_{14}\text{H}_{28}$) into its constituent components; n-heptane (C_7H_{16}) and toluene (C_7H_8). Its activation energy (E_f) is switched between two values, one for conventional combustion and one for HCCI combustion. This is due to that the same model constants could not reproduce a reasonable fit for both modes when compared to e.g. the experimental data published in [13]. This modeling approach and recent modifications to the DOS model are described in depth in [14].

The second global reaction of Table 1 is included to save computational time, since it simulates a number of elementary reactions. If this reaction is omitted, the mechanism will not predict the heat release well due to a lack of HCO radicals. The third and fourth global reactions represent simplified models of soot inception. The oxidation of soot particles by O_2 molecules is described by the Nagle and Strickland-Constable, NSC, model [15]. However, OH radicals are the primary soot-oxidizing agent.

Soot formation

Since aromatic compounds are known to comprise ~10-30% of the constituents of Diesel oil, toluene is included in the DOS model. Aliphatic compounds do not form soot at temperatures below 1800 K, but aromatics do [16]. The low temperature route goes through toluene oxidation, generating the phenyl radical, C_6H_5 . A pathway is included from the phenyl radical to acenaphthylene, A2R5, via the HACA-mechanism (H-Abstraction- C_2H_2 -Addition). In order to avoid having to continue to model the formation of the large polycyclic rings that coalesce to form soot, the elementary reactions stop at the formation of A2R5. The transformation from gaseous soot precursors to solid soot particles (nucleation) occurs via the global graphitization reactions listed in Table 1 (numbers 3 and 4).

NO_x formation

Thermal NO formation is accounted for by the extended Zeldovich mechanism (reactions 1-3 in Table 2). N_2O is formed (reactions 4-7 in Table 2), and can continue to form NO (reaction 8 in Table 2). The formation of NO_2 , which is known occur almost exclusively via the NO molecule mediated by reaction number 9 in Table 2 [17], is also accounted for (reactions 9-12). The Fenimore mechanism, relevant to rich combustion, is not included.

Table 2. Reactions of NO_x (forward reaction rate: $k_f = A_f T^n \exp(-E_f/RT)$ in cm^3 , mole, sec and cal units)

					A_f	n_f	E_f
Extended Zeldovich							
1.	N	+ NO	=	N ₂ + O	3.50E+13	0	330
2.	N	+ O ₂	=	NO + O	2.65E+12	0	6400
3.	N	+ OH	=	NO + H	7.33E+13	0	1120
N₂O formation and destruction							
4.	N ₂ O	+ O	=	N ₂ + O ₂	1.40E+12	0	10810
5.	N ₂ O	+ H	=	N ₂ + OH	4.40E+14	0	18880
6.	N ₂ O	+ OH	=	N ₂ + HO ₂	2.00E+12	0	21060
7.	N ₂ O	+ M	=	N ₂ + O + M	1.30E+11	0	59620
N₂O to NO							
8.	N ₂ O	+ O	=	NO + NO	2.90E+13	0	23150
NO to NO₂ (low E_f – i.e. at low temperatures)							
Main NO to NO ₂ branch							
9.	NO	+ HO ₂	=	NO ₂ + OH	2.11E+12	0	-480
10.	NO	+ O + M	=	NO ₂ + M	1.06E+20	-1.41	0
11.	NO ₂	+ O	=	NO + O ₂	3.90E+12	0	-240
12.	NO ₂	+ H	=	NO + OH	1.32E+14	0	360

THE REFERENCE SPECIES TECHNIQUE

The computational times when including detailed chemical kinetics for diesel fuel into CFD calculations on typical 3D engine meshes, are excessive using today's computers. Computational times of ~1 year were reported in [18] using a single processor, if including 200 species and 100 000 computational cells. Therefore, the treatment of the chemistry must be computationally effective or parallel computing must be used. In the work underlying this thesis, effective treatment is realized using a semi-implicit scheme for the chemical reactions, using the Reference Species Technique, RST, with a small penalty in accuracy.

The species that is in deficiency for each chemical reaction (i.e. closest to being driven to zero concentration when a chemical reaction proceeds) is identified. This specie will then be the reference specie. The reference specie is utilized to linearize the chemical source term, thus avoiding iterations in the treatment of the chemical

kinetics in CFD procedure and hence saving computational time.

In such an approach, the reference species is the only variable, with its concentration taken at the present time step, $n+1$. The reaction partners' concentrations would be constants, taken from the previous time step, n . If $[C_1]$ is the reference species, the chemical production rate of species $[C_i]$ will be:

$$\frac{d[C_1]^{n+1}}{dt} = \sum_{r=1}^R (b_{r1} - a_{r1}) \left(k_f^{r,n} [C_1]^{a_{1r,n+1}} [C_2]^{a_{2r,n}} \dots [C_s]^{a_{sr,n}} - k_b^{r,n} [C_1]^{b_{1r,n}} [C_2]^{b_{2r,n}} \dots [C_s]^{b_{sr,n}} \right)$$

The above equation can be solved analytically, and the reaction progress variable found. In this approach the reactions are solved sequentially, although updating the species concentration after each reaction. Hence, the point at which a reaction is placed in the mechanism will affect the result, which is a source of in-accuracy.

TURBULENCE-CHEMISTRY INTERACTIONS

The Partially Stirred Reactor (PaSR) model has been used in various studies at Chalmers in recent years to predict the effects of sub-grid spatial mixing on the chemical source term. The change in concentration of species s due to one chemical reaction is given by:

$$\frac{d[C_s]}{dt} = \frac{\tau_{chem}}{\tau_{chem} + \tau_{mix}} \left(\frac{d[C_s]}{dt} \right)_{Arrhenius}$$

The solution for changes in the concentration of species s over all reactions is not coupled, but solved sequentially, although updating the species concentration after each reaction.

The shortest chemical time, τ_{chem} , is calculated using the Arrhenius reaction rate, k_f^r , and the species in excess, C_i , according to:

$$\tau_{chem} = \frac{1}{k_f^r [C_s]}$$

The mixing time is calculated using the $k - \varepsilon$ turbulence model according to:

$$\tau_{mix} = C_{mix} \frac{k}{\varepsilon}$$

C_{mix} is a constant, set to 0.015 and 0, for conventional and HCCI combustion, respectively. For HCCI combustion, with C_{mix} was set to zero, the PaSR model

is abandoned and $\frac{d[C_s]}{dt} = \left(\frac{d[C_s]}{dt} \right)_{Arrhenius}$. This

approach i.e. assuming the mixture fully homogeneous within each computational cell, have also been reported by the Lawrence Livermore National Laboratory (LLNL) [19] and the Engine Research Center at the University of

Wisconsin (ERC) [20] for premixed combustion. In other publications by ERC, a model for turbulence-chemistry interactions which is very similar to the PaSR model has been developed and used for HCCI combustion, i.e. their Kong-Reitz model [21].

SPRAY BREAK-UP

The Kelvin-Helmholtz, K-H, and Rayleigh-Taylor, R-T, models are implemented in a competing manner, i.e. the droplet breaks up by the mechanism that predicts the shortest break up time.

The K-H instabilities are due to turbulence within the fuel, leading to perturbations at the liquid/gas interface, whereas the R-T instabilities mainly occur as a result of deceleration of the droplets due to drag. The change in radius of the parent drop is given by:

$$\frac{dr_0}{dt} = -\frac{r_0 - r_d}{\tau}$$

Where r_d is the radius of the child drop and τ is the break-up time. For the K-H instabilities the break-up time is:

$$\tau_{KH} = 3.726 B_1 \frac{r_0}{\Lambda \Omega}$$

where Λ is the unstable wavelength on the liquid surface and Ω is the frequency. A lower value of B_1 represents more disturbances. Values of B_1 in the literature range from 1.73 to 60. In this article, a value of 40 is used. After wall impingement turbulence increases and B_1 is switched to another value (1.73), since Reitz et al. found that this value yielded good agreement with experiments [22].

For the R-T instabilities the break-up time is:

$$\tau_{RT} = \frac{C_3}{\Omega}$$

where the constant C_3 usually ranges between 0.1 and 5.33 [23].

Close to the orifice, R-T is usually the governing mechanism (due to the high velocity of the spray here, which subsequently decelerates), while the K-H breakup becomes more dominant further downstream. In order to be able to reproduce experimentally obtained intact core or breakup lengths, e.g. those given by the correlation:

$$L_{bu} = C \cdot \sqrt{\frac{\rho_f}{\rho_g}} \cdot d_{noz}$$

the R-T model, which would predict extremely rapid breakup directly at the nozzle exit, is "switched off" within this breakup length. The constant C usually ranges from 1 to 1.9 [24].

Prior to conducting the CFD simulations presented in this paper modeled spray penetration lengths were compared, for validation purposes, empirical data

presented by Sandia [25]. The experiments were conducted in an inert environment, in a high-pressure/high-temperature vessel, containing only stoichiometric combustion products with back-pressures of 43 and 113 bar, and injection pressures of 540 and 1630 bar, respectively.

Figure 3 and Figure 4 show the results for two different break-up length constants (C), illustrating that the best fit must be a compromise in order to ensure reasonable agreement for both high back/injection pressure (113 bar and 1630 bar, respectively) and the lower back/injection pressure (43 bar and 540 bar, respectively) cases.

The constants used for the simulations that generated the data illustrated in Figure 4 were used for all further CFD simulations, as summarized in Table 3.

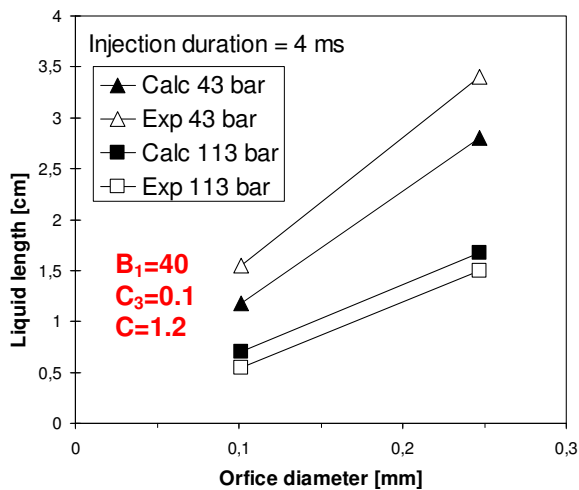


Figure 3. Experimental and calculated liquid core lengths when the break-up length constant, C , (see page 4) is set to 1.2

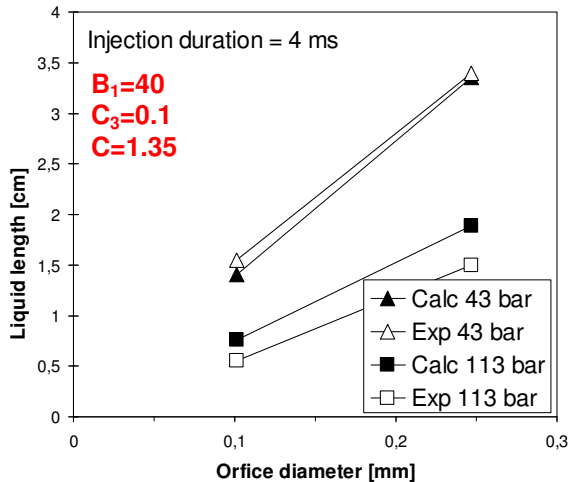


Figure 4 Experimental and calculated liquid core lengths when the break-up length constant, C , (see page 4) is set to 1.35

Table 3. Parameters of the spray/break-up models

K-H Breakup Constant (B_1 before wall impingement)	40
K-H Breakup Constant (B_1 after wall impingement)	1.73
R-T Breakup Constant (C_3)	0.1
Breakup Length Constant (C , distant)	1.35
Initial Sauter Mean Radius	0.8 of the nozzle radius
Initial Sauter Mean Radius Distribution	Constant size
Number of Injected Parcels	2000
Spray Cone Angle [degree]	12.5

ENGINE CHARACTERISTICS AND COMPUTATIONAL GRID

FREE PISTON VS. CRANKSHAFT-DRIVEN MOTION

Figure 5 presents a comparison of the piston motion of the FPE and a corresponding crankshaft-driven engine. The most pronounced difference is that for the FPE the piston is closer to TDC during the whole compression stroke. Then, just after TDC the expansion is more rapid,

which is favorable for the $P \frac{dV}{d\theta}$ term, since it coincides

with the highest pressure period. The maximum free piston speed is lower than for a conventional engine with the same rotational speed/frequency and stroke, but the peak value is reached more quickly.

Although the FPE does not employ a crank mechanism, virtual CAD notation will be used to indicate timings hereafter, since it is widely used. However, the quoted CADs are merely time equivalents calculated as $CAD = time \times frequency \times 360$ ($frequency = rpm/60$).

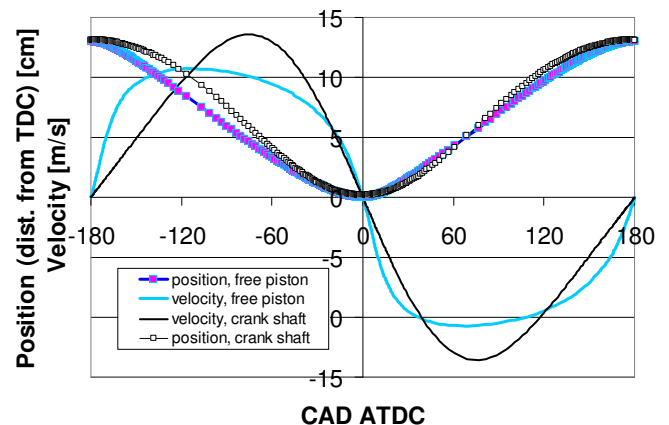


Figure 5. Predicted free piston position and velocity compared to a corresponding crank-shaft driven engine at 31.96 Hz/1918 rpm

ENGINE CONFIGURATION

In order to obtain the best possible scavenging a uniflow scavenging system is used. Twelve inlet ports with a swirl angle of 20 degrees are evenly spaced around the circumference at the bottom end of the cylinder and four pneumatically driven exhaust valves are situated in the cylinder head. The boost pressure for scavenging is generated by an external compressor. The main engine characteristics are listed in Table 4.

The piston bowl design is shallow, since it is designed for HCCI combustion. For the late injections associated with conventional diesel combustion, special caution must be taken in order to avoid spray impingement on the piston.

The KIVA-3V code calculates the injection velocity using specified inputs for orifice area, injection duration and injected mass. The injection velocity, v , is commonly approximated by:

$$V = c_D \sqrt{\frac{2\Delta P}{\rho_f}}, \text{ where } c_D \text{ is the discharge coefficient, } \Delta P$$

is the pressure drop across the nozzle and ρ_f is the fuel density. The required pressure drop across the nozzle was estimated by solving for ΔP .

Figure 6 shows the relationship between injection pressure and velocity when assuming a backpressure of 50 bar. The maximum provided injection rail pressure is 1600 bar, which corresponds to spray velocities of ~483 m/s, if the injector is not choked (choking was not considered). In addition, the rail pressure is assumed to be identical to the sac pressure.

Table 5 presents the characteristics of the two different electromagnetic injectors available for use in the prototype engine. Although the HCCI-relevant injector has more orifices, it does not compensate for the smaller diameter, and a longer injection duration is needed to obtain the same injection velocity and injected mass (see Figure 7).

Table 4. Main engine characteristics

Bore [cm]	10.2
Maximum stroke length [cm]	14
Number of exhaust valves	4
Number of inlet ports	12
Ports height [cm]	2.2
Swirl angle [deg]	20
Height - cylinder head to start of ports [cm]	11.5
Bowl volume [m ³]	12.95*10 ⁻⁶
Valve recession volume [m ³]	13.3*10 ⁻⁶
Intake boost pressure [bar]	1.23 (1.3)
Maximum injection pressure [bar]	1600

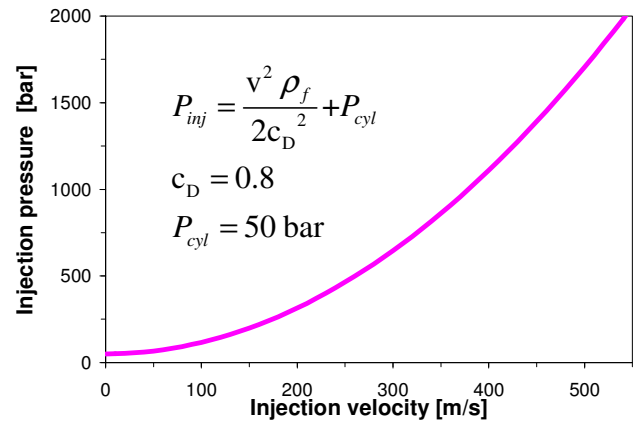


Figure 6. Injection velocity vs. injection pressure

Table 5. Injector specifications

Injector	HCCI Combustion Relevant	Conventional Combustion Relevant
Included angle [deg]	60	140
Nozzle orifice diameter [mm]	0.113	0.158
Number of nozzle orifices	8	5

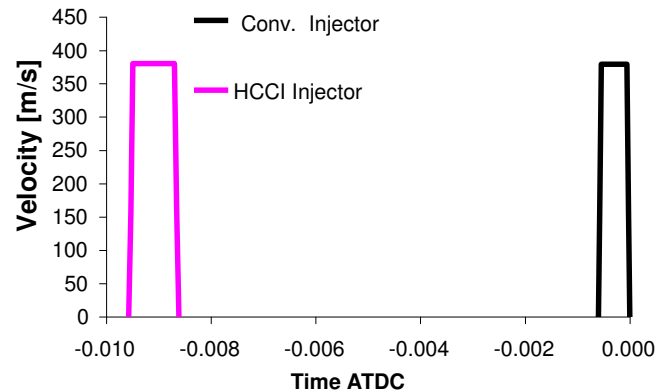


Figure 7. Injection velocity vs. time plots for the two injectors, injecting 17 mg of fuel with the same injection pressure

OPERATING CONDITIONS

The EGR level is calculated according to:

$$EGR\% = \frac{CO_{2in}}{CO_{2out}} \cdot 100.$$

Five different compression ratio (r_c) cases were investigated in this thesis, using the procedure described in the previous section. The r_c is reduced by lowering the reference kinetic energy of the piston at the last sensor position. After the reference kinetic energy was selected it was kept constant and an iterative procedure was used until convergence. The results are shown in Table 6.

Reducing the r_c leads to a reduction in frequency if having similar combustion phasing (e.g. compare case

A-B / case C-D of Table 6), since it reduces the in-cylinder pressure force.

When a variable such as the Start Of Injection (SOI) or injection pressure is changed, the consequent changes in the ROHR will cause the piston motion to slightly change, as illustrated in Figure 8. Here, retarding the SOI causes the piston to easily move to a position closer to TDC (higher r_c) at the same time as the combustion is phased later, causing lower peak in-cylinder pressures and therefore lower frequencies.

These interactions complicate attempts to analyze the effect of one variable on others, e.g. SOI on fuel consumption and emissions, since they also depend on r_c and frequency. Therefore, the piston position curve was kept constant when comparing different cases, i.e. the iterative procedure was not used, except for cases A-E. An identical r_c can also be obtained through iterations, if adjusting the reference kinetic energy accordingly for each new case (this was not made here). Then, only the frequencies will differ from the previous iteration.

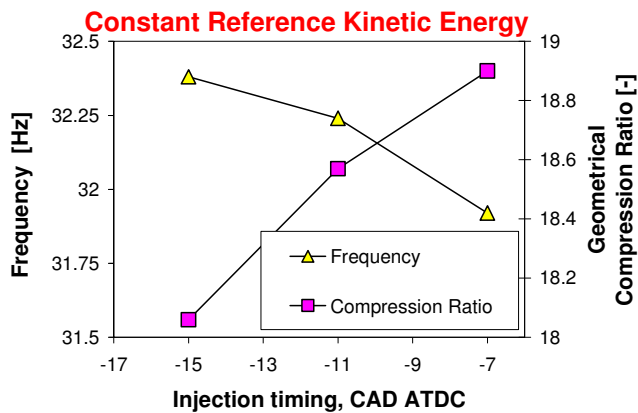


Figure 8. Piston motion characteristics vs. SOI for case B, if using a constant reference kinetic energy

Table 6. Operating conditions

Case	A	B	C	D	E
Combustion mode	Conv.	Conv.	HCCI	HCCI	HCCI
Compression ratio [-]	23.23	18.5	16.3	14.7	13.8
Frequency [Hz]	34.8	32.2	32.6	31.6	31.2
Corresp. rotational speed [rpm]	2090	1933	1957	1898	1870
Kinetic energy - last sensor position [J]	338	250	235	200	180
SOI [CAD ATDC]	-11	-11	-100	-100	-100
Compressor power [kW]	1.35	1.35	1.35	1.35	1.35
Injection pressure [bar]	1510	1010	1000	1000	1000
Corrected squish [cm]	0.465	0.615	0.714	0.800	0.858
Stroke [cm]	13.395	13.095	12.898	12.724	12.610
Injected fuel mass/stroke [mg]	17	17	17	17	17
Internal and external EGR [%]	~15	~15	55	55	55

COMPUTATIONAL GRIDS

A 1/5th geometrical wedge including only one of the spray jets was used to save computational time, while retaining good resolution of the spray (see Figure 9). The computational mesh displayed in Figure 9 features nearly 140 000 cells at BDC. It was used for conventional combustion, while for HCCI combustion, in which an 8-orifice nozzle was applied, a 1/8th geometrical wedge was used. The prototype engine has valve recessions, which are not included in the computational grid. However, their volume is accounted for in the dead volume.

Multicycle simulations, including the open and closed part of the cycle, were conducted using a full 360-degree grid, including inlet ports and exhaust valves (see Figure 10).

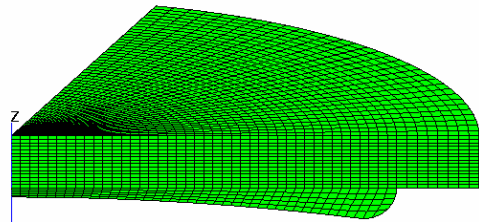


Figure 9. The 1/5:th geometrical wedge grid at TDC

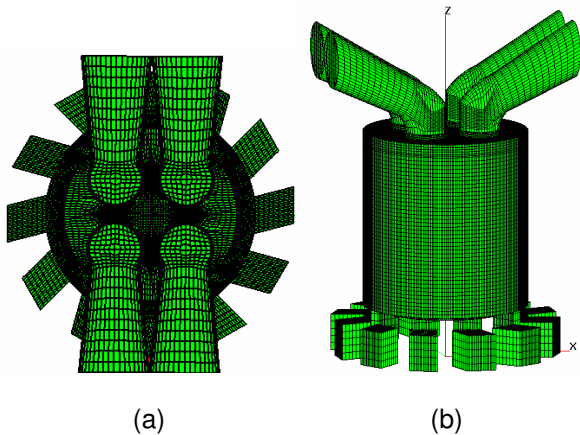


Figure 10. The full 360-degree grid displayed from (a) the top and (b) the side

DETERMINATION OF THE FREE PISTON MOTION

The piston motion is predicted using a MATLAB/SIMULINK model, developed within the EU Growth Project G3RD-CT-2002-00814. This model employs Newton's second law, accounting for the forces acting on the piston/translator, i.e. friction forces, electrical forces, and in-cylinder gas forces. A detailed description of the physical models included can be found in [26].

IMPROVED PHYSICAL MODELS

The original simplified models for ignition and combustion were not used. Instead, the ROHR and ignition delay for each case was taken from the CFD/detailed chemistry simulation. In addition, a model for the complete engine (constructed in GT-POWER) was used to determine the temperature and pressure during the open part of the cycle, up to Exhaust Valve Closing (EVC).

Other models for in-cylinder processes were also improved, e.g. instead of using a constant value for the ratio of specific heats (γ), it was calculated using NASA polynomials in the GT-POWER code (see Figure 11a). Two average values characteristic of the compression and expansion strokes were chosen and exported to the MATLAB/SIMULINK model. Figure 11a and Figure 11b show γ and specific heat (C_p) values as functions of time (CAD ATDC) for:

1. A mixture with no EGR and early injection (90 CAD BTDC)
2. A mixture with EGR and late injection (5 CAD BTDC)
3. A mixture with EGR and early injection (90 CAD BTDC)

In order to isolate the effect of mixture composition (e.g. with no temperature effects) on the specific heat, a predetermined, identical ROHR curve was applied to all cases. The results show that the presence of EGR in the mixture has very little effect on the specific heat and γ .

However, these parameters were considerably reduced by the presence of fuel vapor (see the case with SOI at 90 BTDC).

THE CONTROL SYSTEM

The model was recently updated to include a control system [27], which is considered the most vital part of the FPE. In the prototype engine, the translator position is measured using evenly spaced sensors. The electrical force is updated during the stroke so that a reference kinetic energy is met at the position of the last sensor, where the electrical machine is switched off.

Identical control algorithms were implemented in a MATLAB/SIMULINK model to those used in the prototype engine. Figure 13a shows how the electrical force is varied during one engine cycle, Figure 13b shows the generated power and Figure 14a shows the position and pressure during engine start-up, before fuel injection. For the first strokes the valves are closed, at a later stage the valves are opened and at the last stage injection starts.

THE ITERATIVE PROCEDURE

Since the piston motion and frequency, the rate of heat release and the initial in-cylinder conditions all affect each other, while predicted using different modeling tools with no direct coupling between them, an iterative procedure was used. An illustration of the modeling tools can be found in Figure 12.

An example of the final iterative stage is shown in Figure 14b in the form of pressure traces obtained from the CFD code and the MATLAB/SIMULINK model.

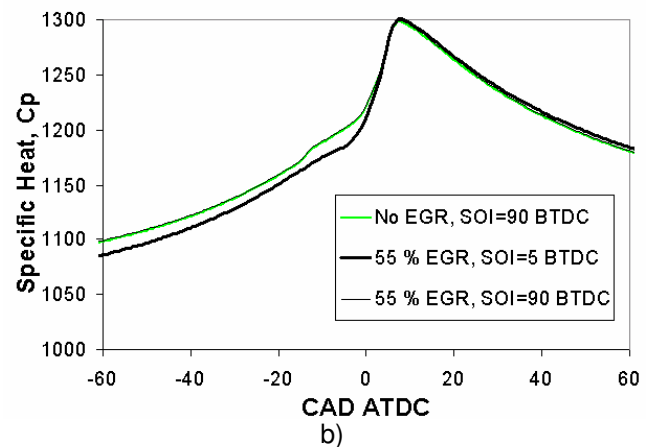
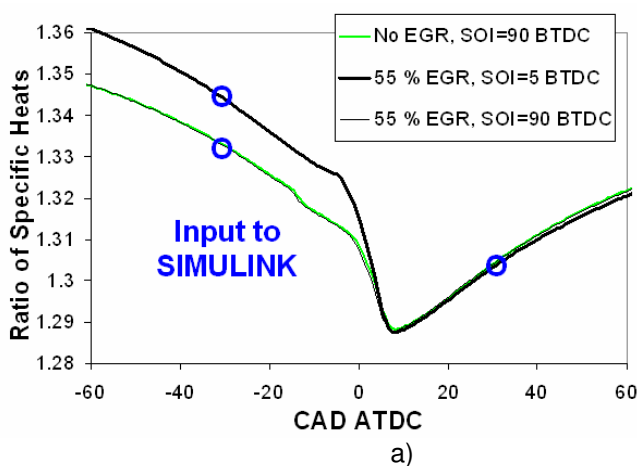


Figure 11. (a) The ratio of specific heats (γ), calculated using NASA polynomials and provided as input to the MATLAB/SIMULINK model (one value for the compression and one for the expansion stroke) and (b) the specific heat vs. CAD