

Establishing New Correlations Between In-Cylinder Charge Motion and Combustion Process in Gasoline Engines Through a Numerical DOE

2010-01-0349

Published
04/12/2010O. Laget, J.M. Zaccardi and X. Gautrot
IFPT. Mansion
RenaultE. Cotte
PSA

Copyright © 2010 SAE International

ABSTRACT

This paper presents an innovative methodology and the corresponding results of a study whose goal is to identify the main links between in-cylinder charge motion and the development of combustion without taking into consideration how to create this charge motion (shape of the intake ducts, valve timing, *etc.*...).

During this study a specific methodology was developed and used. It is based on the calculation of a “3D numerical test bench” matrix planned following the Design Of Experiments method. Many aerodynamic configurations obtained by combining the three main aerodynamic motions with several different intensities (tumble, cross-tumble or swirl) at the intake valve closing were calculated.

The results obtained allow on the one hand to validate and to extend some correlations between in-cylinder charge motion and combustion found in the literature and on the other hand to highlight new links between macroscopic aerodynamics and the quantities related to the combustion velocity, timing and efficiency.

INTRODUCTION

The drive for substantial CO₂ reductions in the light of the Kyoto Protocol and higher fuel efficiencies have increased research into gasoline engines. Optimal geometry design

including appropriate combustion chamber and intake/exhaust ducts shapes is required. In order to design appropriate intake ducts and valve timings, the optimum in-cylinder air flow giving the optimum combustion efficiency must be defined.

The goal of the present study was to improve the understanding of the effects of the in-cylinder flow field on the combustion. To do so, new correlations between characteristics of in-cylinder charge motion and combustion evolution were established.

The effects of in-cylinder charge motion on combustion have been investigated for many years and an extensive literature is available on this subject. More than 500 references have thus been perused within the framework of this project to draw up a report on the techniques allowing the characterization of the in-cylinder flow field and also to list the most interesting studies giving some correlations between in-cylinder charge motion and combustion for different problems like the resistance to knock or the acceptance of dilution. For more than 20 years, the analysis of the effects of the charge motion has been a major asset for the development of spark ignition engines since it represents one of the main sources of cyclic fluctuations penalizing their global efficiency.

This problem has already been experimentally tackled with several optical diagnoses like LDV¹, PIV or DGV. These methods allow to precisely analyze the generation and the evolution of the aerodynamic field inside the combustion chamber. It is thus possible to associate these motions to classical engine tests to identify the effects of the in-cylinder charge motion on combustion. This process is however time-consuming since it is first necessary to characterize different configurations on a steady state flow bench or on a transparent engine before manufacturing some prototypes and to test them on a test bench. More recently, the considerable developments of calculation codes have helped accelerate this identification and the whole process which consists of optimizing the design of the combustion chamber and the intake ports thereby allowing a fast and efficient comparison of numerous very different configurations.

Despite all these efforts, some information are still lacking and the design of the intake ducts and the optimization of the compromise between volumetric efficiency and charge motion are still mainly guided by specific skills and experiences gathered by each car manufacturer. Besides, it should also be noted that an important gap seems to exist between aerodynamics experts and combustion experts.

The search of links between aerodynamics and combustion during this present study was performed on two different engines running at different loads. In this paper, only the first case corresponding to a downsized boosted engine is presented. Indeed, the knock problem was taken into account to develop the methodology described in this paper.

The 3D computations aims at testing different aerodynamic motions (some of them being hard or even impossible to be experimentally realized). Indeed, after a reference computation used to fix several initial and boundary conditions in terms of pressure and temperature (validated by comparisons with experimental data), several variations of the charge motion at the intake valve closing were performed. The imposed motions for each specific aerodynamics were built using a composition of the three main aerodynamic motions (tumble, cross-tumble and swirl, see [appendix](#) for the definition of aerodynamic motions). These different configurations were planned according to the Design Of Experiment method allowing the reduction of the number of computer runs over a traditional gridding technique, an optimized post processing of the results through the fitting of polynomial models and an in depth research of the different criteria linking aerodynamics to combustion.

Relevant correlations were defined using two different approaches. First of all, links between the local aerodynamics characteristics (in time and space) and the combustion

analysis results were identified. Secondly, these local aerodynamics data were correlated to the global air flow characteristics (main motions such as tumble, cross-tumble, swirl). Finally, the direct link between global air flow and combustion could be deduced.

NUMERICAL SETUP

INTRODUCTION

Since a large amount of aerodynamic conditions at intake valve closing (IVC) need to be tested, some of which are not easily realizable with conventional intake ducts, a numerical test-bed was relied on. It allows to save time by avoiding the conception phase of the intake ducts (the desired aerodynamics are imposed at IVC) and provides a large amount of data which would have needed advanced and intrusive experimental devices to be experimentally obtained (local and fast measurements of pressure, temperature and velocity for example).

The present study was performed with the IFP-C3D three-dimensional Computational Fluid Dynamics code. IFP-C3D is developed and maintained at IFP and takes benefit from the latest physical model evolutions allowing to solve, in unstructured hexahedral meshes, the unsteady turbulent Navier-Stokes equations for chemically reactive gaseous mixtures.

DESCRIPTION OF THE CFD CODE

The three-dimensional simulations were performed using IFP-C3D [1], [2]. This one solves the unsteady equations of a chemically reactive mixture of gases, coupled with the equations for a multi-component vaporizing fuel spray. This 3D CFD parallel (Open MP and MPI) code solves the Navier-Stokes equations using an ALE-extended finite volume method on conformal unstructured hexahedral meshes (Arbitrary Lagrangian Eulerian, see [3], [4] and [5] for more details concerning this model and its validation). The code uses the well known time splitting decomposition. The temporal integration scheme is largely implicit.

For turbulent combustion, the **3-Zone Extended Coherent Flame Model (ECFM3Z)** is used [6]. The auto-ignition prediction and the influence of chemistry on reaction rates are provided by the TKI auto-ignition model [7], shortly described below.

Ignition and combustion modeling

In general for gasoline engines the auto-ignition is not systematically taken into-account. Nevertheless, during the present program, one of the computed set points was chosen in order to study the influence of the charge motion on the

¹Acronyms are defined in section DEFINITIONS/ABBREVIATIONS

knock phenomenon. Consequently two kinds of ignition must be modeled (auto-ignition which is at the origin of knock and spark ignition) joined to the representation of the combustion (diffusion or propagation flame). It is why two kinds of ignition model are used in the present study: the first one, AKTIM to model the ignition assisted by a spark plug and the second one, TKI for the auto-ignition.

The combustion model must be adapted to the two kinds of ignition appearing in such charged SI engine subject to the knock phenomenon. It is why the unified ECFM3Z model was used. This one describes the turbulent mixing in each mesh cell and provides the local thermodynamical conditions and species concentrations to the auto-ignition model.

After the start of combustion, this model represents the chemical kinetics related to the combustion process and is well suited to define the local conditions in order to well represent the combustion progress

AKTIM: Spark plug ignition model

The Ark **K**ernel **T**racking **I**gnition **M**odel (AKTIM) [6], [8], is used to describe the flame kernel expansion in gasoline SI engine. This model is divided into four complementary sub-models. Indeed it allows to take into account the different characteristics of the secondary electrical circuit (initial electric energy, resistance, and inductance) as well as the spark plug's geometry. The latest is modeled by discrete particles so as to avoid the remeshing of the combustion chamber in order to take the plug into account. The spark itself is modeled by particles informally placed along the spark path. The spark shape and size is function of the flow field surrounding the plug. Thus, this model is well adapted to take into-account the influence of charge motion inside the combustion chamber on the ignition.

The ECFM 3Z Model

For the present case of study a combustion process which is a blend of two elementary combustion modes has to be taken into account. Indeed the combustion model must be able to well represent the chemically-controlled auto-ignition (knock phenomenon occurring in charged SI engine and to be avoided) and subsequent rapid oxidation (premixed combustion) as well as the mixing-controlled diffusion flame. The ECFM3Z model (more complete description available in [8]) allows to correctly represent the local mixing process occurring in the combustion chamber before the auto-ignition during and after the combustion

The TKI Model

The TKI (**T**abulated **K**inetics of **I**gnition) model (more extensively described in [7]) is used to compute the auto-ignition. It relies on tabulated auto-ignition quantities issued from detailed chemistry calculations performed with the Senkin code (part of the Chemkin package). Databases have

been built for a wide range of thermodynamic conditions representative of cool flame ignition as well as main ignition using a detailed mechanism (416 species, 1992 reactions) for different n-heptane/iso-octane mixtures issued from the DCPR laboratory (Département de Chimie Physique des Réactions) at the University of Nancy. The 3D model variables are calculated once and for all for a large range of initial thermodynamic conditions and are stored in lookup tables. The fresh (or unburned) gas quantities that are used as input parameters of the auto-ignition lookup tables are the temperature, the pressure, the fuel-air equivalence ratio, the rate of dilution and the octane number.

ENGINE CONFIGURATION, INITIAL AND BOUNDARY CONDITIONS

The geometry of the combustion chamber studied (see INTRODUCTION section) is shown in [Figure 1](#). The following table gives the different characteristics of the engine, operating point and meshes used for the computations.

Table 1. Main characteristics of the engine, operating point and meshes used for the computations

Data	Value
Bore (mm)	69
Stroke (mm)	80.5
Connecting rod length (mm)	139
Compression Ratio	9.5
Engine speed (rpm)	2000
IMEP (bar)	15
Spark Advance (Cad)	13
λ	1
EGR ratio (%)	0
Nb Hexa @ IVC	303554
Nb Hexa @ TDC	129842

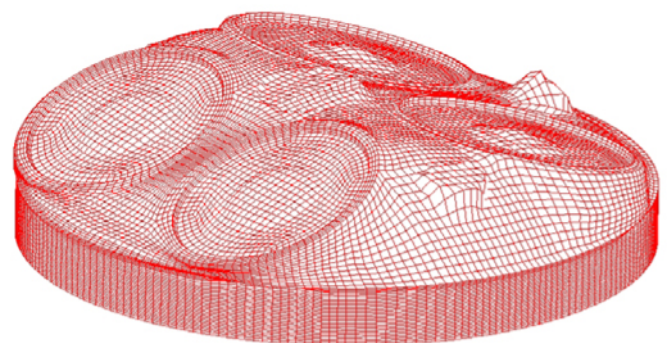


Figure 1. Mesh of the combustion chamber of the engine

Computations are started at IVC. The initial flow fields are composed of the three main rotational motions centered on the X, Y and Z axes (respectively called cross-tumble, tumble and swirl, see [Figure 20](#)) of various intensity. The reference point used as origin of the resulting rotational motion axis was also varied.

The initial and boundary conditions were deduced from the reference computations. These computations were done in order to reproduce experimental measurements with the same methodology as applied in the rest of the study. The intensity of the in-cylinder charge motion of these reference calculations was used to define the reference charge motion of the DOE points computed.

For the engine case presented here, a computation of the intake stroke showed that for the reference operating point the charge motion induced by the intake ducts geometry can be directly approached by a pure tumble motion of intensity 1.3 at the intake valve closing.

The initial (at IVC) and boundary conditions used for these reference computations are given in the following table:

Table 2. Initial and boundary conditions for the reference computations

Data	Value
IVC (CAD BTDC)	153
Temp. @ IVC (K)	384
Press. @ IVC (KPa)	141.5
Tumble @ IVC	1.3
Piston Temp. (K)	470
Head Temp. (K)	440
Liner Temp. (K)	425
Inlet Valve Temp. (K)	440
Exh. Valve Temp. (K)	440

METHODOLOGY

PREPROCESSING

In order to find links between in-cylinder charge motion and combustion, several aerodynamic configurations have to be tested. In order to realize these tests rapidly on an operating point representative of a given engine context and in order to set up a Design Of Experiments (DOE), a numerical test bed was used. The use of CFD allows to compute a wide range of complex aerodynamic configurations without dealing with time-consuming design and manufacturing issues. The computations were realized with the following general conditions:

- in order to minimize the computational time, only the compression and the beginning of the expansion strokes were

computed (until 50 CAD ATDC). The term IMEP refers thus only to this period within the cycle;

- the air-fuel ratio and the temperature of the mixture were supposed to be homogeneous in the combustion chamber (the injection was not modeled and was supposed to occur early in the engine cycle);
- finally, for each of the different aerodynamics inducing different combustions, the spark advance had to be optimized according to a criterion specific to each studied running condition. Indeed, for the present engine (downsized high load) the spark advance of each configuration had to be chosen in order to run the engine just at the knock limit. The choice of the spark advance for all computed configurations required numerous computations (the total amount of calculations is 195 for this engine).

HYPOTHESES OF COMPUTATIONS

During this study, only a part of the engine cycle was modeled. This particularity induces some limitations of the methodology. Indeed, only the compression and a part of the expansion strokes were computed (computations initialized at IVC and stopped 50 CAD). Thus, the evolutions of volumetric efficiency as a function of the intake pipe and valve topologies are not taken into account (differences in terms of heat exchange and pressure drop). The fields of temperature and Fuel/Air equivalence ratio are supposed homogeneous for all the different tested aerodynamics (the injection was not modeled). Consequently, the induced differences in terms of combustion efficiency between each configuration can not be well represented. Moreover, concerning the exhaust stroke, the influence of the charge motion modifications is not taken into account. The alterations of the emptying of the combustion chamber may induce evolutions in terms of dilution due to trapped burnt gases but these phenomena were not supposed to be very influential on charge motion.

Imposing the initial conditions (other than velocity) as well as boundary conditions identical for all the studied cases probably reduces the gap between the different aerodynamic configurations proposed. For example, the heat exchange measurements are distorted due to the fact that the wall temperatures are the same whatever the aerodynamics and as a consequence whatever the considered combustion. Moreover, the spark plug is modeled using a Lagrangian object without taking into account the heat exchanges and the boundary layer effects.

The RANS (Reynolds Average Navier-Stokes) equations solved during the computations allow to represent only a mean engine cycle. Consequently the problem of cycle to cycle variability or combustion stability cannot be properly modeled.

All these limits were well identified at the beginning of the project but do not constitute obstacles. Indeed, the methodology was simplified through several assumptions to isolate first the effects of charge motion. It could be then efficiently completed in order to take into account more parameters (such as EGR, wall temperatures, *etc.*...) and computing a larger part of the engine cycle for example.

DEFINING THE KNOCK LIMIT SPARK ADVANCE

For the downsized high load configuration considered here the knock limit spark advance has to be defined for each computed charge motions. In order to do that a criterion was set on the reference case which was experimentally known to be tuned at the knock limit.

The computation of the reference point was performed with the experimental spark advance. The total charge consumption can be numerically divided into the part due to auto-ignition (knock) and the part due to the conventional flame propagation. For the reference point the computation shows a very small amount of fuel consumed by auto-ignition which is consistent with the fact that the experimental data were recorded at the knock limit. Increasing the spark advance (6 CAD) induces an increase of the amount of fuel consumed by the auto-ignition (see [Figure 2](#)). A more extended Spark Advance (SA) variation was performed on this operating point. [Figure 3](#) showing the time evolution of the charge consumption due to the auto-ignition for several SA. The criterion for the knock limit spark advance is based on the evolution of the ratio between the sum of the charge consumption due to auto-ignition and that due to conventional flame propagation. This ratio is calculated for all the computed spark advances and all the charge motion configurations and is normalized by the reference one (reference charge motion and experimental SA).

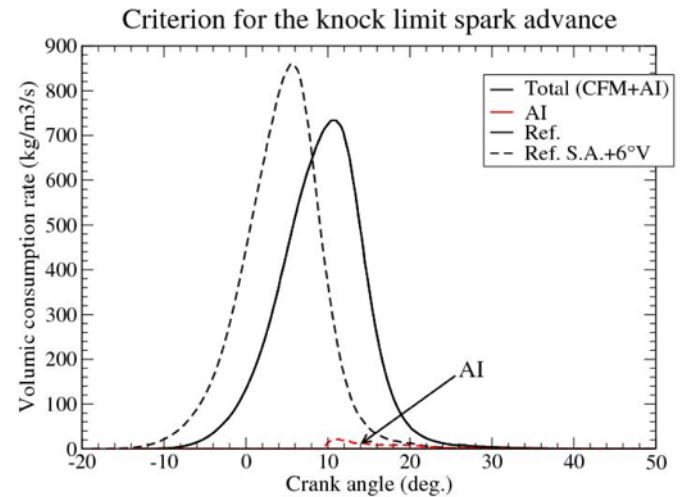


Figure 2. Total and auto-ignition charge consumption for experimental spark advance and for an increased one (6 CAD)

Thus this ratio (critAI) is a function of the spark advance. The criterion used to define the knock limit spark advance is given below.

$$\text{critAI} = \frac{\int \omega_{\text{AI}}}{\int \omega_{\text{Tot}}} \bigg/ \left(\frac{\int \omega_{\text{AI}}}{\int \omega_{\text{Tot}}} \right)_{\text{Ref}}$$

- critAI > 1: knock
- critAI < 1: too low SA
- critAI = 1: knock limit SA

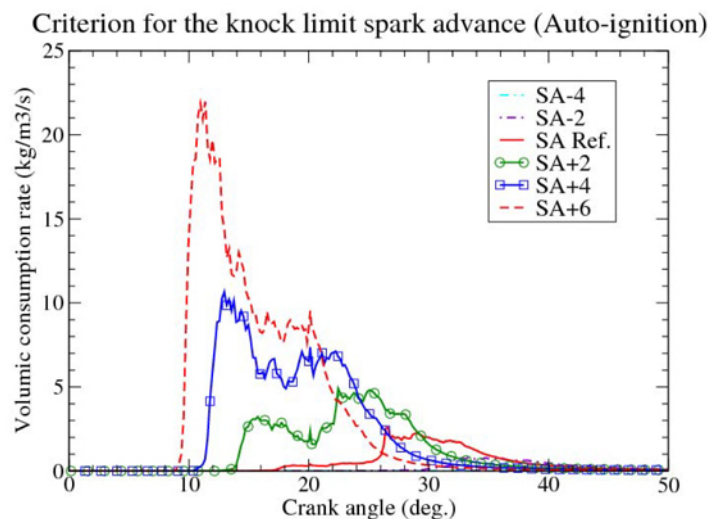


Figure 3. Evolution of the auto-ignition charge consumption function of spark advance

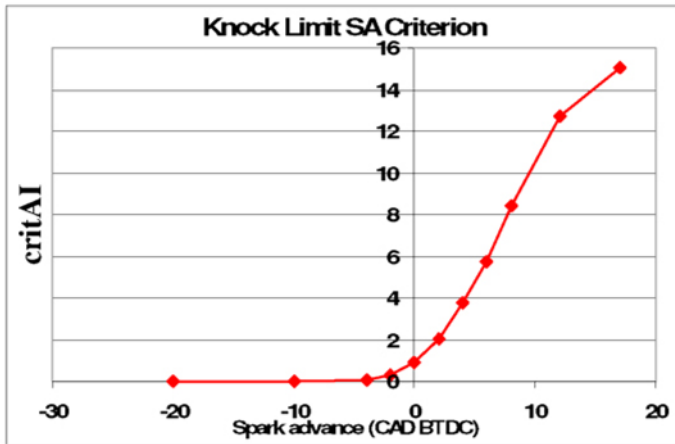


Figure 4. Evolution of the critAI ratio function of the SA (according to the experimental SA)

DESIGN OF EXPERIMENTS

Assumptions

Several assumptions were made to build the Design Of Experiments:

- variations of the flow field are performed via the initialization of the global velocity field at IVC;
- the secondary aerodynamic motions (Ω -swirl et Ω -tumble, see [appendix](#) for the definition and [Figure 21](#)) are not taken into account. Only the main motions (swirl, tumble and cross-tumble) are used to build the initial aerodynamic motions;
- a single initial rotation center is considered for all the motions;
- tumble motion should remain the main motion regarding spark ignition engines;
- and finally, an energetic concept is integrated into our DOE to take into account the energy that the intake path line would give to the air flow.

Consequently, the following input parameters were chosen to build the D-optimal DOE (see [appendix](#) for precision concerning aerodynamic motions and centre of rotation):

- TR^2 : square of the tumble motion intensity;
- SR^2 : square of the swirl motion intensity;
- E : parameter which can be assimilated to an energy equal to the sum of the squares of the three aerodynamic main motion intensities ($E = TR^2 + SR^2 + CR^2$). The intensity of the cross-tumble motion can be directly deduced from this data and the two previous ones;
- X_{Centre} : relative abscissa coordinate (according to the centre of combustion chamber) of the origin of the initial rotation axis;

- Z_{Centre} : vertical relative coordinate (according to the centre of combustion chamber) of the origin of the initial rotation axis in the $Y=0$ plane.

The parameter Y_{centre} has not been taken into account since it has been estimated that the tumble motion was the main motion to consider for a first approach. It also had the advantage of limiting the number of parameters and thus the calculation time.

Thus, only variations concerning the initial or boundary conditions between each configuration are performed on the topology, the intensity or/and the origin of the rotational aerodynamic initial motion imposed in the cylinder at the IVC (start of computation).

DOE Structure

A D-Optimal DOE for quadratic models with second-order interactions was chosen to fit the model's parameters and an other Space-Filling DOE was made up to validate the obtained model (see [9] for more details about the DOE method and D-optimal DOE). The choice a priori of this type of DOE was conditioned by mathematical and physical reflections. A global trade-off was considered between the physical representations of inputs, their narrowed spaces of variation and the fact that the evolutions of the main targeted outputs (turbulent intensity, mean velocity, etc.) function of the inputs were not supposed to follow higher orders polynomials. It should be noted that the choice of the DOE depends on the form of problems that have to be tackled. Other kinds of DOE and models could have been used but it has been decided to use a classical sort to set up a methodology, to apply it with only five simple parameters and to qualitatively validate it.

The computation sequence is the gathering of a training set consisting in 26 runs (namely 125% of the number of terms in the canonical form of the model) and a validation set consisting in 7 runs planned in order to maximize the distance between them and the 26 training runs distributed in the domain (space filling technique). Moreover, 3 more training points were added in order to get a low energy ($E = TR^2 + SR^2 + CR^2$) level (due to the fact that one of the 5 possible energy levels was not used by the first planned points). Finally, contrary to classical test campaign performed on a physical engine, the repeatability was not assessed since talking about repeatability has no sense with RANS calculations.

To sum up, 36 tests were planned. The different parameters of the DOE are quoted above. It may be specified that the range of variation of each parameter was limited. Especially for the E parameter (similar to an energy) since a minimal value was defined reachable only using a minimal tumble