

Investigation on the Potential of Quantitatively Predicting CCV in DI-SI Engines by Using a One-Dimensional CFD Physical Modeling Approach: Focus on Charge Dilution and In-Cylinder Aerodynamics Intensity

Alessio Dulbecco, Stephane Richard, and Christian Angelberger
IFP Energies Nouvelles

ABSTRACT

Increasingly restrictive emission standards and CO₂ targets drive the need for innovative engine architectures that satisfy the design constraints in terms of performance, emissions and drivability. Downsizing is one major trend for Spark-Ignition (SI) engines. For downsized SI engines, the increased boost levels and compression ratios may lead to a higher propensity of abnormal combustions. Thus increased levels of Exhaust Gas Recirculation (EGR) are used in order to limit the appearance of knock and super-knock. The drawback of high EGR rates is the increased tendency for Cycle-to-Cycle Variations (CCV) it engenders. A possible way to reduce CCV could be the generation of an increased in-cylinder turbulence to accelerate the combustion process. To manage all these aspects, 1D simulators are increasingly used. Accordingly, adapted modeling approaches must be developed to deal with all the relevant physics impacting combustion and pollutant emissions formation. In this study, a CCV modeling approach for system simulation integrated into the 1D gas CFM1D combustion model has been used to reproduce CCV in a downsized Direct Injection (DI) - SI single cylinder engine. The CCV model was developed by integrating physical understanding gained from 3D CFD based on a Large-Eddy Simulation (LES) approach. The experimental database includes a complete engine map, single-parameter variations of dilution rate as well as standard and increased aerodynamics operating conditions. Once the CCV model was calibrated, it was applied to simulate CCV over the complete experimental database and showed a good reproduction of experimental observations. As a result of using a physics based CCV model, it was possible to acquire some understanding of the reasons for the observed combustion variability from the performed simulations. Finally, with the perspective of using the model for engine control applications, its potential to run in Real Time (RT) was evaluated.

CITATION: Dulbecco, A., Richard, S., and Angelberger, C., "Investigation on the Potential of Quantitatively Predicting CCV in DI-SI Engines by Using a One-Dimensional CFD Physical Modeling Approach: Focus on Charge Dilution and In-Cylinder Aerodynamics Intensity," *SAE Int. J. Engines* 8(5):2015, doi:10.4271/2015-24-2401.

INTRODUCTION

The requirement to further improve the efficiency of SI engines while at the same time aiming at near-zero pollutant emissions motivates new technology developments; recent trends for SI engine architectures identify the downsizing technology as a promising approach. The related increased boost levels and compression ratios amplify however the appearance of abnormal combustions. Accordingly, to avoid knock and super-knock, dilution of the in-cylinder fresh charge with EGR is required. The drawback of using high EGR rates is the destabilization of the combustion process, leading to an increase of the in-cylinder pressure CCV. This contributes to limit the range of operation of such technology. Accordingly, the understanding of CCV is becoming an important topic of research [1, 2, 3, 4, 5, 6]. A promising way to reduce CCV could be the generation of an increased in-cylinder turbulence at Intake Valve Closing (IVC) to accelerate the combustion process [7]. To deal with CCV, different simulation approaches can be considered depending on the scope.

Control algorithms are increasingly developed by using 1D CFD engine simulators running close to RT [8, 9]. However, at present such simulators cannot account for some of the major sources of CCV. Although they naturally reproduce CCV related to the unsteady flow in the intake, exhaust and fuel injection systems, sources associated to the non-linear interactions of the turbulent flow inside the cylinder and combustion processes cannot be addressed. An approach often used to model CCV resulting from combustion-amplified stochastic turbulence effects is to perform multi-cycle simulations in which the input parameters of standard 1D-CFD combustion chamber models are randomly perturbed from cycle to cycle [10, 11, 12, 13, 14, 15, 16]. The resulting 1D-CFD models are able to reproduce some features of the experimentally observed CCV, but a shortcoming is their incapability to predict the system behavior for interpolated/extrapolated operating conditions. Indeed the statistical distribution and amplitude of the perturbations to be introduced in each cycle, as well as the choice of the variable to

perturb, are rather the result of a fitting procedure of experimental cyclic cylinder pressure variations, than based on physical grounds. This is directly due to the lack of a detailed understanding of the basic chain of events leading to the appearance of CCV. Such approaches help identifying some major reasons for CCV for specific engines and operating points, but cannot in general be used to predict the onset of CCV outside of the domain their parameters have been determined for. Another possible approach to reproduce stochastic CCV using 1D CFD combustion chamber models is based on Monte-Carlo methods to resolve the joint probability of the considered in-cylinder flow variables. Such stochastic combustion chamber models allow reproducing some of the effects leading to CCV, namely stratification of the fresh mixture or spatial variations of cylinder wall temperatures [17]. Although such approaches have an interesting potential to reproduce CCV, they suffer from important computational time requirements and do hardly allow accounting for effects related to turbulence or resolved convection, which may have to be introduced using a random perturbation like described above. Finally the predictions of such models strongly depend on the numerical parameters used to solve the joint probability density function, making it hard to distinguish “physical” CCV from those arising from the numerical resolution of the underlying equations.

At the other end of the spectrum of simulation techniques, 3D-CFD has received increasing attention for studying CCV, due to the unique possibilities offered by the LES technique [1,2,18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30]. Recent studies have specifically explored the potential of LES for studying the sources of CCV inside the combustion chamber of Port Fuel Injection (PFI) SI engines [1,23,24]. They also illustrated how analyzing this LES allows identifying sources for CCV. In the studied case, they were found to be stochastic in nature and related to the turbulence of the intake flow and its coupling to SI and flame propagation. Similar studies were reported in [27,30] to study CCV related to diluted operation, while [18] performed a LES of a DI-SI engine in order to study the interaction between the flow generated during intake and liquid fuel spray.

In this study, the CCV modeling approach for system simulation integrated into the 12 gas CFM1D combustion model [31], developed at IFP Energies nouvelles (IFPEN), has been used to reproduce CCV of a downsized DI-SI single cylinder engine, which has been run at IFPEN in the framework of the French research project ICAMDAC. The CCV model was developed by integrating physical understanding gained from 3D CFD LES. Because of the physical bases of the approach, the model can manage the physical aspects impacting the combustion process and pollutant emissions formation which are relevant in phase of new engine design and control definition strategies, allowing to reduce the costs of the development and the time to market of the new engine. Moreover for this same reason, it is possible to acquire from the simulation results some understanding of the reasons of an eventual combustion process destabilization. The experimental database includes a complete

engine map, single-parameter variations of dilution rate as well as standard and increased aerodynamics¹ operating conditions. Once the CCV model calibrated, it has been used to predict CCV over the complete experimental database. The model showed a high potential in CCV prediction. According to the observed results, a calibration methodology is then proposed to allow a direct industrial application of the model. The CFM1D-CCV results were then compared to LES CCV results in order to validate the fact that the proposed calibration is representative and consistent with physical findings for a given set of operating conditions. Finally, with the perspective of using the model for engine control applications, its potential to run in Real Time (RT) was tested.

MODELING APPROACH

The approach used in this work is based on the CFM1D combustion model integrating CCV [31].

The CFM1D Combustion Model

The CFM1D combustion model, [32, 33, 34], is dedicated to the flamelet regime and is well adapted to premixed and partially premixed combustion processes, which represent the main oxidation mechanisms in SI engines. The basic equations of the model were obtained by reduction of the 3D CFD CFM model [35].

The CFM formalism distinguishes two zones: fresh and burned gases, which are separated by a flame front propagating from the burned gases towards the fresh mixture. Chemical reactions of fuel oxidation occur in the flame front, a very thin layer compared to all scales of the turbulent flow, and post-flame chemistry takes place in the burned gases.

Heat Release Calculation

The enthalpy balance for the combustion chamber is:

$$\frac{dmh}{dt} = V \frac{dP}{dt} + \frac{dQ_{comb}}{dt} - \frac{dQ_{wall}}{dt} + \sum_{in,out} h_j \frac{dm_j}{dt} \quad (1)$$

where V is the cylinder volume, h is the mass specific enthalpy, m is the enclosed mass and P is the pressure which is linked to the mean temperature T by the perfect gas law. The last RHS term of Equation 1 refers to enthalpy exchanges at the system inlets and outlets (valves, injector). Q_{wall} corresponds to heat losses at the walls which are described by the Woschni model [36]. The combustion heat release, Q_{comb} , is computed by using the following differential equation:

$$\frac{dQ_{comb}}{dt} = \sum_i h_{f_i} \left(\left. \frac{dm_i}{dt} \right|_{ff} + \left. \frac{dm_i}{dt} \right|_{pf} \right) \quad (2)$$

1. Standard and increased aerodynamics conditions represent different levels of turbulence intensity. Both operating conditions were characterized by means of flow bench experiments.

where h_{f_i} is the formation enthalpy of species i , $\left. \frac{dm_i}{dt} \right|_{ff}$ and $\left. \frac{dm_i}{dt} \right|_{pf}$

are the mass variations of this species respectively in the flame front and in the burned gases due to post flame chemistry reactions,

respectively. $\left. \frac{dm_i}{dt} \right|_{ff}$ can be expressed as a function of the fuel consumption rate:

$$\left. \frac{dm_i}{dt} \right|_{ff} = v_i \frac{W_i}{W_{fuel}} \left. \frac{dm_{fuel}}{dt} \right|_{ff} \quad (3)$$

where v_i is the stoichiometric coefficient of the gas i in the reaction of fuel oxidation through the flame and W_i denotes the molar weight of this gas. The fuel consumption rate is then given by a flame

propagation model described in the next paragraph. Finally, $\left. \frac{dm_i}{dt} \right|_{pf}$

is computed using a reduced chemistry approach [32] accounting for CO and NO formation in the burned gas zone.

Flame Propagation Model

The consumption rate of fuel throughout the flame front depends on the turbulent flame surface St and the fresh gases properties according to:

$$\left. \frac{dm_{fuel}}{dt} \right|_{ff} = \left. \frac{dm_{fuel}^{fg}}{dt} \right|_{ff} = -\rho_{fg} Y_{fuel}^{fg} U_l S_t \quad (4)$$

Where ρ_{fg} is the fresh gases density, $Y_{fuel}^{fg} = \frac{m_{fuel}^{fg}}{m_{fg}}$ is the fuel mass

fraction in the fresh gases and U_l is the laminar flame speed computed by using correlations depending on pressure, fresh gases and temperature (here is used a correlation for gasoline [32]). The turbulent flame surface S_t is written as the product of a mean surface S_m and the flame front wrinkling factor Ξ :

$$S_t = \Xi S_m \quad (5)$$

The mean surface is computed considering a spherical expansion of the flame from the spark plug to the cylinder walls in a realistic geometry. For this purpose, S_m is a priori tabulated as a function of the burned gases volume and the piston position as described in [32] or [37]. The flame wrinkling factor evolution is given by a physical equation describing the progressive transition from the laminar kernel to the fully turbulent flame. This equation was obtained by reduction of the 3D equation for the flame surface density [24,35] and includes the unsteady effect of the turbulent flow on the flame stretch through the efficiency function Γ proposed in [38]:

$$\frac{1}{\Xi} \frac{d\Xi}{dt} = \Gamma \left(\frac{u'}{U_l}, \frac{l_t}{\delta_l} \right) \frac{u'}{l_t} \left(\frac{\Xi_{equ} - \Xi}{\Xi_{equ} - 1} \right) - \frac{2}{r_{bg}} (1 + \tau) (\Xi - 1) U_l \quad (6)$$

u' is the instantaneous velocity fluctuation, l_t is the integral length scale, δ_l is the laminar flame thickness estimated using Blint's

correlation [39], $\tau = \frac{\rho_{fg}}{\rho_{bg}}$ is the thermal expansion rate,

$r_{bg} = \left(\frac{3V_{bg}}{4\pi} \right)^{\frac{1}{3}}$ is the current mean radius of burnt gases and Ξ_{equ}

corresponds to the Ξ value when an equilibrium is reached between turbulence and flame wrinkling Ξ [24]. The main unknowns of the problem finally correspond the turbulent flow field properties u' and l_t .

Turbulence Model

The integral length scale is assumed to be close to the piston-cylinder head distance D , as proposed by many authors [36,40], while

$u' = \sqrt{\frac{2}{3}k}$ is obtained from a classical two equation model

describing the evolution of the mean and turbulent kinetic energy densities, K and k , respectively [32]. This model accounts for energy transfers from the mean flow into turbulence during the compression stroke and for dissipation of k at the Kolmogorov scale. Such a 0D model cannot be entirely predictive as the complex flow field evolution strongly depends on the engine geometry and is here not directly resolved as in 3D CFD calculations. Accordingly, some parameters of the turbulence model were adjusted following the procedure described in [32] allowing to generate maps for the tumble ratio and the integral length scale, and kept constant in the following part of the study.

CCV Modeling in CFM1D

The system simulation CCV approach was developed at IFPEN in the framework of the LESSCCV European R&D project [31]. This approach accounts for CCV related to in-cylinder aerodynamics, turbulence properties and mixture composition. Their impact was identified as playing a major role in the appearance of CCV based on LES [31]. To improve the prediction of the very first instants of combustion a detailed spark-ignition model accounting for the electric circuit properties has been included, in order to be able to set the right initial conditions originating from the plasma created between the spark electrodes. The CCV model adopts a statistical approach to describe cycle-to-cycle fluctuations of tumble and the integral length scale by means of Gaussian Probability Density Functions (PDF); practically, for each engine cycle, values of tumble and integral length scale are randomly extracted from their respective PDFs and used to initialize the turbulent flow field within the cylinder. These properties of the flow play a very important role

on the development of the combustion process, computed by the CFM1D model, at all levels: going from the spark ignition time, at which the velocity field bends the initial kernel of burnt gases formed between the electrodes, to the laminar/turbulent transition of the flame front and the wrinkling of the flame surface. The impact of the mixture composition (equivalence ratio or dilution) on combustion and consequently on CCV is naturally accounted for in CFM1D through the laminar flame speed and the wrinkling efficiency function.

While the parameterization of the detailed ignition model is straightforward, as it depends on known geometrical/electrical quantities related to the ignition electric-circuit (such as inter-electrode distance or coil inductance), two parameters have to be specified to calibrate the standard deviations of the Gaussian PDFs of tumble and integral length scale, respectively σ_{tumble} and σ_{l_t} at IVC:

$$\sigma_{tumble} = \frac{std(TUMBLE)}{mean(TUMBLE)} \cdot 100 \quad (7)$$

in which

$$std(TUMBLE) = \sqrt{\frac{1}{N-1} \cdot \sum_{i=1}^N (TUMBLE_i - mean(TUMBLE))^2} \quad (8)$$

and

$$\sigma_{l_t} = \frac{std(l_t)}{mean(l_t)} \cdot 100 \quad (9)$$

in which

$$std(l_t) = \sqrt{\frac{1}{N-1} \cdot \sum_{i=1}^N (l_{t_i} - mean(l_t))^2} \quad (10)$$

N is the number of considered engine cycles.

Parametric Study of the CCV Model

In this section different parameterizations of the CFM1D-CCV model were tested and compared in order to show the model sensitivity to variations of the CCV model parameters, σ_{tumble} and σ_{l_t} . Figure 1 shows four “basic calibrations” whose role is to highlight the impact of a given parameter on the computed results, independently of the other.

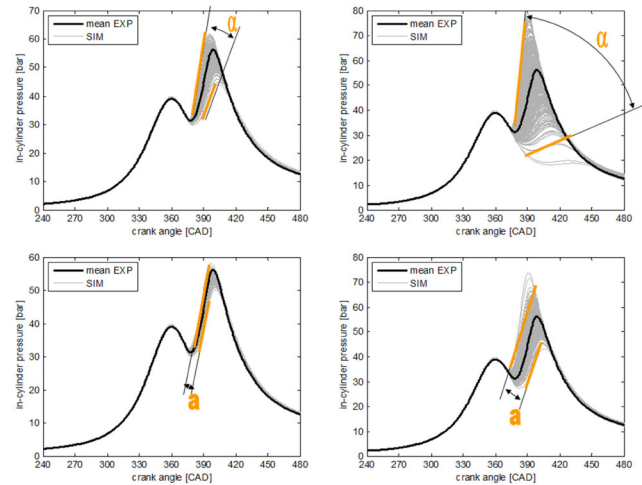


Figure 1. computed CCV results (engine operating conditions: 19 bar, 1800 rpm, standard tumble and 0% of dilution) given by the CF1D-CCV model relative to four different parameterization: $\sigma_{tumble}=5\%$; $\sigma_{l_t}=0\%$ (upper-left corner), $\sigma_{tumble}=17\%$; $\sigma_{l_t}=0\%$ (upper-right corner), $\sigma_{tumble}=0\%$; $\sigma_{l_t}=5\%$ (lower-left corner) and $\sigma_{tumble}=0\%$; $\sigma_{l_t}=17\%$ (lower-right corner).

The two parameters, σ_{tumble} and σ_{l_t} , do not have the same impact on the result. Specifically, by observing the in-cylinder pressure envelopes, it can be noticed that:

- an increase of σ_{tumble} , upper part of Figure 1, produces an increase of the angle α , characterizing the in-cylinder pressure envelope spread. This is coherent with the fact that a wider range of tumble number values implies a wider range of turbulence levels within the cylinder and, consequently, a wider range of combustion speeds; the combustion speed determining at a first order the slope of the in-cylinder pressure due to combustion,
- an increase of σ_{l_t} , lower part of Figure 1, produces an increase of the in-cylinder pressure envelop spread identified by the linear distance a . This is coherent with the fact that a wider range of integral length scale values implies a wider range of instants at which the flame transitions from laminar to turbulent propagation regime, for a given engine operating condition. The transition to turbulent combustion regime increases considerably the Heat Release Rate (HRR) in agreement with the level of turbulence within the cylinder, which is kept constant ($\sigma_{tumble}=0$) and which, in turn, determines to first order the slope of the in-cylinder pressure increase due to combustion.

At the light of the considerations presented above about the influence of the calibration parameters on the results, it is possible to identify on the morphology of the in-cylinder pressure envelopes the contributions of the two parameters.

Statistical Convergence

As a preliminary step, the statistical convergence of the results was verified. In fact, in order to have reliable results, for given engine operating conditions, an appropriate minimum number of engine cycles must be used to compute the instability variables. In the context of the ICAMDAC project, it was experimentally observed

that, while for an Engine Operating Point (EOP) belonging to the engine map 300 cycles are enough, for a diluted EOP the convergence requires 500 cycles. Accordingly, simulations with different numbers of engine cycles were performed for diluted EOPs as illustrated in Figure 2. These simulations clearly show that 111 cycles are not enough to obtain converged statistics, while by using 500 cycles (number of cycles retained for experimental converged results) the results are almost converged; in fact, by passing from 500 to 1111 cycles the difference is negligible. It is noteworthy that through the numerical simulation it was possible to get the same kind of information than that obtained with experiments; namely:

- to obtain converged statistics of EOP relative to “unstable operating conditions” (presence of knock, high dilution rates) at least 500 cycles are necessary,
- the number of cycles required for statistical convergence increases with the dilution rate.

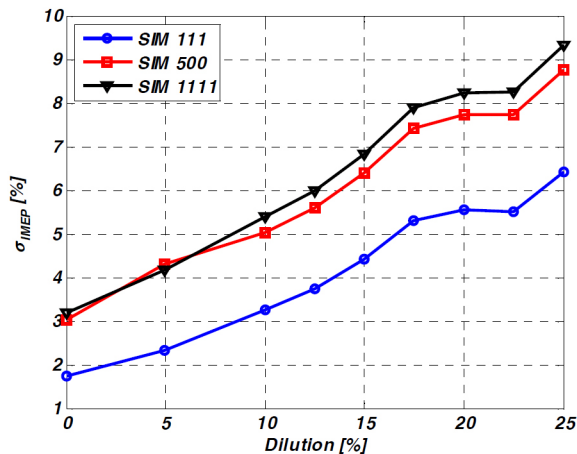


Figure 2. σ_{IMEP} (EOP: 19 bar, 1800 rpm, standard tumble) as a function of the dilution rate for different numbers of simulated cycles.

For what follows, in order to have the most reliable comparison between experimental and simulated results, the number of cycles computed is set to that of the experiments.

EXPERIMENTAL DATABASE

The experimental database produced in the framework of the ANR project ICAMDAC has been selected to evaluate the capability of the CFM1D-CCV model for system simulation to quantitatively predict the CCV observed experimentally; the database refers to a single-cylinder DI-SI engine architecture, Table 1. The database considered, was generated at IFPEN with the aim of generating and understanding abnormal combustion processes such as knock and super-knock. It includes the engine map and several elementary structured single-parameter variations (e.g. Spark Advance (SA), Start Of Injection (SOI), dilution rate, in-cylinder aerodynamic intensity, trapped air, spark ignition-energy, fuel type).

In the context of this study, a reduced domain of EOPs including the engine map and single parameter variations of intake gas dilution rate for two different levels of in-cylinder aerodynamics intensity was retained.

The dilution which aimed at being representative of the impact of EGR, was carried out by using pure N_2 in order to avoid the impact on the combustion process of specific compositions of the dilution gases. The dilution rate, DIL, is defined as:

$$DIL = \frac{\dot{m}_{N_2}}{\dot{m}_{air} + \dot{m}_{N_2}} \cdot 100 \quad (11)$$

where $\dot{m}_{air}$ and $\dot{m}_{N_2}$ represent the mass flow rates of air and N_2 , respectively, at the intake. Figure 3 represents the considered engine operating domain; the dilution variations at nominal and increased tumble refer to the EOP characterized by 19 bar of Indicated Mean Effective Pressure (IMEP) and 1800 rpm of engine speed on the engine map, for which the presence of knock has been recorded.

Table 1. Experimental single-cylinder engine specifications.

Displaced volume	399.5 cc
Stroke	85.8 mm
Bore	77 mm
Connecting Rod	132.2 mm
Compression ratio	10.5:1
Number of Valves	4
Exhaust Valve Open	43° BBDC @ 0.7 mm lift
Exhaust Valve Close	6° ATDC @ 0.7 mm lift
Inlet Valve Open	7° BTDC @ 0.7 mm lift
Inlet Valve Close	24° ABDC @ 0.7 mm lift
Fuel	Issoctane
Injector	Continental, 6 holes

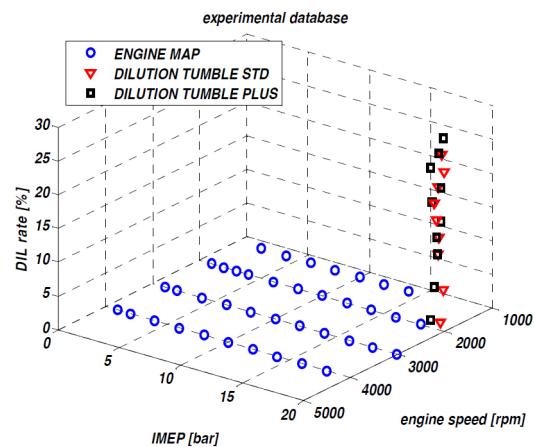


Figure 3. Experimental engine operating domain. It includes load variations from 1 to 20 bar of IMEP, engine speed variations from 1200 to 4000 rpm and dilution rate variations from 0 to 27.5 %.

The CCVs on the experimental engine maps described in terms of $IMEP_{inst}$, $P_{max_{inst}}$ and $\alpha_{p_{max_{inst}}}$, Figure 4, Figure 5, Figure 6, are respectively defined as:

$$IMEP_{inst} = \frac{std(IMEP)}{mean(IMEP)} \cdot 100 \quad (12)$$

$$P_{max_{inst}} = \frac{std(P_{max})}{mean(P_{max})} \cdot 100 \quad (13)$$

$$\alpha_{p_{max_{inst}}} = \frac{std(\alpha_{p_{max}})}{mean(\alpha_{p_{max}})} \cdot 100 \quad (14)$$

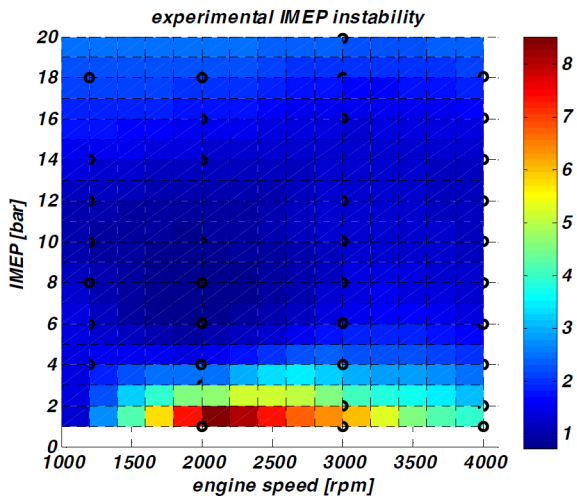


Figure 4. Experimental IMEP instability over the whole engine map.

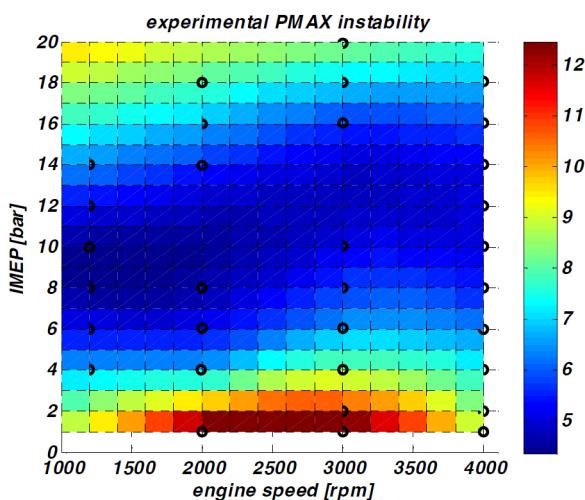


Figure 5. Experimental maximum-pressure instability over the whole engine map.

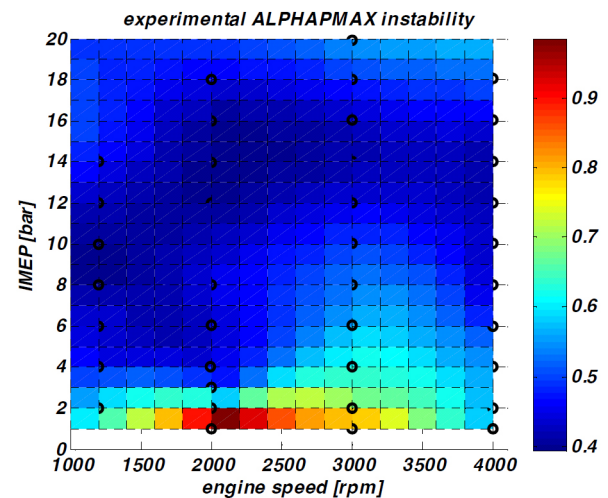


Figure 6. Experimental maximum-pressure angle instability over the whole engine map.

The evaluation of P_{max} and $\alpha_{p_{max}}$ instabilities are computed by accounting for the maximum in-cylinder pressure peak related to combustion and not the absolute in-cylinder pressure peak, Figure 7.

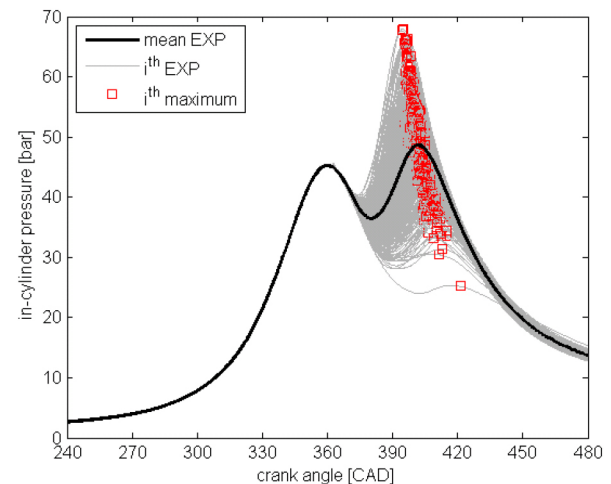


Figure 7. experimental in-cylinder pressure traces (EOP: 19 bar, 1800 rpm, 15% DIL, standard tumble). Valuable information for CCV estimation are mean pressure curve, pressure curve envelop and maximum pressure distribution in terms of P_{max} and αP_{max} .

NUMERICAL SIMULATION

This section is dedicated to the calibration of the system simulation CCV model in order to reproduce the CCVs of the experimental engine map described above. Concerning the engine map, the number of computed engine cycles per EOP is 300, while for diluted EOPs the number of computed engine cycles is 500.

Engine Simulator

An Amesim simulator of the ICAMDAC single-cylinder engine was built to represent the experimental facility, Figure 8. The CFM1D-CCV model was calibrated in two steps: first, combustion was calibrated in order to fit the mean in-cylinder pressure curve of the experimental database; during this calibration phase the CCV model