Due to recent improvements on combustion modeling, three-dimensional Computational Fluid Dynamics (CFD) tools have become essential in aiding in the engine development. They allow the understanding of basic combustion phenomena and help to optimize various parameters of a pre-existing combustion system or to design a combustion chamber from scratch. Combustion systems become more and more complex in order to address the conflicting demands based on more stringent emission standards and customer desire for “fun-to-drive” vehicles, i.e. higher torque and power output. Therefore ignition, combustion and pollutant models are constantly evolving in order to keep CFD in pace with engine innovations. Such a close follow-up of IFP’s CFD tools enables 3D computations to contribute actively in all IFP engine research projects.

This paper describes the contribution of 3D combustion modeling to the development of gasoline engines, from conventional PFI or DISI turbo-charging (downsizing with power increase) to high power output (Formula One) applications.

INTRODUCTION

CFD tools are able to reply to increasing engine development needs as the analysis they provide helps to improve our understanding and shorten/lower development time/cost. This is amplified with the increase of CPU power and the democratisation of “low cost” parallel super-scalar machines. In order to fully benefit from such a market evolution, IFP has developed a new solver [1] IFP-C3D, using a hexahedral, unstructured and parallel (OpenMP [3]) formalism, dedicated to internal combustion engines. Some capabilities of the code have already been presented last year for Diesel applications [2] and we concentrate in this paper on gasoline applications.

IFP-C3D has been used in several projects and been applied to different gasoline configurations for which it has proven useful and accurate.

THE IFP-C3D CODE

IFP-C3D solves the unsteady equations of motion of a chemically reactive mixture of gases, coupled with the equations for a multi-component vaporising fuel spray. The Navier-Stokes equations are solved using a finite volume method extended with the ALE (Arbitrary Lagrangean Eulerian) method. IFP-C3D uses the well known time splitting decomposition. The temporal integration scheme is largely implicit. Concerning liquid phase, evaporation, break-up using the Wave-FIPA model and spray/wall interaction are included. For turbulent combustion, the 3-Zone Extended Coherent Flame Model (ECFM3Z) developed at IFP is used [4]. The Ark kernel tracking ignition model (Aktim) [5] is used for spark ignition modelling.

Species (and tracers), energy and the RNG k-ε turbulent [6] diffusion terms are all implicitly solved by a generic diffusion routine. Moreover, preconditioning efficiently the pressure matrix drastically reduces the simulation time. The convection terms are explicitly sub-cycled. A second order upwind scheme for scalar and momentum convection is used.

The OpenMP paradigm [3] was chosen for parallelization as it is standardised, portable, scalable, adapted to modern super-scalar SMP machines and has an attractive performance to development cost ratio (easy to implement using only compiler directives). Profiling of the sequential version allowed to concentrate paralleling efforts on the most CPU time consuming routines: pressure solver, diffusion, matrix inversion (conjugate gradient method with SOR algorithm) and gradient terms (for pressure and
Reynolds stress tensor). Overall, about 50% of the code is parallelized providing a very good and scalable speed-up (around 3 for 4 processors) on our benchmark cases. Details and extensive validation of the code can be found in [2].

An automatic mesher is available in IFP-C3D to create 2D/3D unstructured sector meshes with periodic boundary conditions, simplifying the task of meshing for an automatic use of the code. Figure 1 shows the mesh of a typical \( \omega \) shaped bowl-in-piston configuration as generated with the automatic mesher. For fully 3D geometries IFP-C3D can import meshes generated with a commercial mesher. In particular, for the ANSYS ICEMCFD HEXA grid generator, a direct output is available to create meshes readable by IFP-C3D in the CGNS standard portable format [7]. Moreover, the extensive use of 2D/3D periodic meshes combined with the automatic re-meshing algorithm (to automatically adapt the mesh refinement) drastically reduces the CPU time cost while improving simulation accuracy.

The IFP-C3D code is fully integrated in the AMESim platform [8] with a friendly user interface (c.f. Figure 2), a basic mesh viewer and automatic post-processing. The integration in the AMESim platform is driven by the desire to make 3D simulations simple and give 3D combustion models access to other 1D libraries.

In this way, 3D CFD can be used conjointly with the 1D engine simulation library (IFP-ENGINE) to extend or replace the experimental combustion maps used in the 1D combustion model. For transient engine operation simulation, the 1D/3D coupling enables the user to get valuable combustion data for operating conditions that are not available from experiments [9]. This feature bridges a gap between 3D combustion and 1D system simulations and will be extended in the near future by a direct temporal coupling between IFP-C3D and the IFP-ENGINE library, as well as other 1D libraries (thermal, hydraulic…).

**CONVENTIONAL PFI ENGINE**

In this example, we show the capacities of the combustion models used in the IFP-C3D code to estimate, for a given operating condition, the best spark advance and the evolution of in-cylinder pressure.

The engine used in the study is a small 4 cylinder engine with increased fuel efficiency. The operating condition is 2000 RPM and 6 bar BMEP.
The models used for the simulations are:

- ECFM3Z model for combustion
- Aktim model for ignition (c.f. Figure 6)

The specificity of such simulations is the ability to accurately reproduce ignition and combustion in a very wide range of engine configurations.

In Figure 3 the velocity field in the vicinity of the spark plug can be observed. One should notice that the fluid motion is the result of a complete intake stroke computation. Here, only the results of compression and combustion strokes are discussed.

In Figure 3 the velocity field around the spark plug can be observed. One should notice that the fluid motion is the result of a complete intake stroke computation. Here, only the results of compression and combustion strokes are discussed.

![In-cylinder pressure](image)

**Figure 4: Simulated and experimental in-cylinder pressures**

![Calculated IMEP](image)

**Figure 5: Computed and experimental IMEP**

![Aktim model in action](image)

**Figure 6: Aktim model in action: plug, spark and ignition kernels (A and B) and start of combustion (B)**

As can be seen in Figure 4, the computations are in good agreement with the test bench results in terms of in-cylinder pressure. Using the experimental spark timing in the simulations, it can be observed that the models are able to reproduce quite well the value and the timing of the maximum in-cylinder pressure. Moreover, for the same operating conditions, the numerical in-cylinder pressure signal is very close to the experimental one. Figure 5 shows the evolution of computed IMEP versus spark timing. We should note that the numerical IMEP is evaluated using only part of the high pressure cycle (between 120 crank angle degrees BTDC and 120 degrees ATDC) and cannot be directly compared to the experimental value. In spite of that, it can be concluded that the numerical best spark advance is quite close to the
experimental one. This suggests that the combustion and ignition models are adequate for this type of configuration. IFP-C3D can therefore be used effectively to aid engineers in engine development studies.

**TURBOCHARGED ENGINES**

In recent years the trend to higher fuel efficiency and reduced emissions for the same specific engine output in SI engines has led IFP to focus on turbocharged (i.e. downsized) engines [10]. These goals have been met by use of the direct injection technology and flexible valve timing combined with a twin-scroll turbocharger that allows a larger valve overlap and hence better scavenging of the burnt gases.

The challenge for such boosted engines is to control knock. Therefore the piston bowl has been specifically shaped such that during the valve overlap period optimal scavenging of hot residual gases can be achieved. Direct injection and the cooling effect of evaporating droplets allow in addition to fight knock occurrence while avoiding the short-circuiting of fuel into the exhaust system. In what follows, only the scavenging results are presented which remain a critical part for turbocharged engines.

The investigated engine is a 450 cm³ single cylinder direct injection SI engine with a compression ratio of 10:1. The operating point and the relevant valve timings are listed in Table 1.

<table>
<thead>
<tr>
<th>Engine speed</th>
<th>1250 RPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>BMEP</td>
<td>1.7 MPa</td>
</tr>
</tbody>
</table>

Table 1: Operating point

Figure 7 depicts the scavenging flow in two cutting planes, one slicing through the valves and the other one lying in the symmetry plane, as shown in the lower left corner. The regions coloured in blue mark the intake species, while the red areas show the burnt gases.

At IVO, burnt gases fill out the entire combustion chamber and the exhaust ports as the intake valve lift is yet too small to produce any significant inflow. The instant later at (TDC-20 cad) shows the intake flow which, due to the high pressure difference of around 2 bar between the intake and exhaust port, is rather in horizontal direction. Thus short-circuiting of the intake mixture occurs while efficiently flushing out burnt gases. At EVC residual gases remain largely concentrated in the cylinder head dome.

Figure 8 shows the evolution of the intake and burnt mixture in the combustion chamber, represented by the concentrations of O₂ and CO₂. It is seen that most of the scavenging occurs within the first 25° CA. At EVC the burnt gases comprise around 1% of the mixture, which extrapolated to IVC will result in around 0,1% of residual gases.

The results shown here indicate that a highly efficient scavenging can be achieved with a large valve overlap in twin-scroll turbocharged engines, thus increasing the knock resistance of the engine.
FORMULA ONE ENGINES

A third example concerns Formula One Engines. The aim here is to check the capability of the CFD code to deal with a high power engine, running at high speeds and with a high bore-to-stroke ratio. A methodology has been developed to deal with injection and acoustic waves in the pipes, prior to combustion phase calculations. More details on this methodology can be found in [12]. We concentrate in this paper on the combustion phase, considering only the combustion chamber and removing the pipes. For such calculations, because of the high RPM, the mesh needs to be sufficiently refined to capture all the combustion phenomena, and particularly pressure waves generated by the flame front propagation. The number of cells for the computational domain is about 282,000 at BDC, for half a chamber.

COMPRESSION PHASE: MIXTURE HOMOGENEITY

First of all, the mixture resulting from the intake calculation, is clearly not homogeneous at IVC. The histogram of the mixture strength at IVC is reported in Figure 9. It can be seen that at this stage of the engine cycle, the distribution of equivalence ratio is still quite large with a spread factor of about 3.

COMBUSTION PHASE: RESULTS

Figure 10 shows the computed best spark advance. With the use of the new Aktim ignition model, satisfying agreement with the measured shape is obtained. Indeed the decrease for high spark advance on the right hand side of the curve, is correctly reproduced. The best-calculated spark advance is not exactly equal to the experimental one, although quite close; the calculation gives a higher spark advance than the test bench.

Two possibilities come to mind when trying to explain such a difference. It could be due to the ignition model, which is very sensitive to the range of thermodynamic conditions of racing engines. Indeed, at high RPM (above 18000), 10° CA are equivalent to a few tens of microseconds, and a slight delay in the transition between flame kernel initiation and flame propagation can lead to this order of difference. However, it is also possible that the generation of the IMEP curve from CFD can pick a best timing different from that of experiments, where brake power is determined and best spark advance is given for BMEP instead of IMEP.
This interpretation seems to be supported by the next figure reporting in-cylinder pressures. The 3D calculated pressure curve obtained for experimental best spark advance is in good agreement with the experimental one (Figure 11).

CONCLUSIONS
This paper has shown the flexibility and adaptability of the CFD tools developed at IFP, and how they can be integrated in the Research & Development chain, especially here for gasoline engines. The integration of the IFP-C3D code in the AMESim platform will allow in the near future the total integration of CFD tools in a development/calculation environment from 1D to 3D. The potential of CFD tools is very substantial, since it can be used for a large field of application from "conventional" engines (PFI or using DISI turbo-charging technology) up to the development/improvement of race engines. Moreover, these tools are now sufficiently flexible to be adapted to future engine technologies such as CAI™ engines or using high EGR rates, for which development is currently in progress.

ACKNOWLEDGEMENTS
The authors would like to thank Ferrari for allowing to publish some of these results.

REFERENCES