

3D-CFD Virtual Engine Test Bench of a 1.6 Liter Turbo-Charged GDI-Race-Engine with Focus on Fuel Injection

Marco Chiodi, Antonella Perrone, Paolo Roberti, and Michael Bargende
FKFS

Alessandro Ferrari and Donatus Wichelhaus
VW-Motorsport

ABSTRACT

In the last years motorsport is facing a technical revolution concerning the engine technology in every category, from touring car championships up to the F1. The strategy of the car manufacturers to bring motorsport engine technology closer to mass production one (e.g. turbo-charging, downsizing and direct injection) allows both to reduce development costs and to create a better image and technology transfer by linking motorsport activities to the daily business.

Under these requirements the so-called Global Race Engine (GRE) concept has been introduced, giving the possibility to use one unique engine platform concept as basis for different engine specifications and racing categories.

In order to optimize the performance of this kind of engines, especially due to the highly complex mixture formation mechanisms related to the direct injection, it is nowadays mandatory to resort to reliable 3D-CFD simulations.

In this paper the contribution of intensive CFD simulations within the engine development process will be shown, in particular some issues regarding the injection modeling and its influence on the prediction of the mixture formation, the combustion and the global engine performance will be analyzed.

CITATION: Chiodi, M., Perrone, A., Roberti, P., Bargende, M. et al., "3D-CFD Virtual Engine Test Bench of a 1.6 Liter Turbo-Charged GDI-Race-Engine with Focus on Fuel Injection," *SAE Int. J. Engines* 6(3):2013, doi:10.4271/2013-24-0149.

INTRODUCTION

In the last years, the technical revolution in motorsport powertrain introduced by the international automotive association (FIA: French for Federation International de l'Automobile) in accordance with the major car manufacturers worldwide participating to racing events has the aim to introduce modern engine technology and hybridization solutions which are becoming mandatory for commercial products. In this way motor-racing will get back its role of extreme test bench for technical solutions also used in mass production cars. Concerning the engine it has been decided to use the same platform (called Global Race Engine GRE) for many different racing series. Its main features are: four cylinders, displacement of 1.6 l, DI. This platform can be either naturally aspirated to power a F3 car or overcharged up to 2,5 bar (absolute) for WRC (World Rally Championship) and WTCC (World Touring Car Championship). An exception regards the F1 cars, where a V6 1.6 l DI turbocharged solution has been chosen.

The development of WRC engine from scratch is a very complex task and since its early phases it has been focused on bringing together the best solutions concerning mechanics, tribology, thermodynamics, fluid-dynamics by extensive use of modern simulation tools. In this way, long before the first prototype could be tested at the test bench, different solutions could be compared and selected for the engine optimization. In this paper, after a brief description of the strategies for the global engine layout definition, some particular issues concerning the injection modeling will be analyzed by means of a 3D-CFD tool.

The Role of the Injection Pattern

In order to limit the power output of the engine an air-restrictor of 33 mm placed before the compressor is imposed by the regulations. The other engine specifications are summarized in [Table 1](#); more detailed information can not be provided due to secrecy reasons.

Table 1. Engine specifications

Displaced volume	1597 cc
Bore / Stroke	83 / 73.8[mm]
Compression ratio	>10.5:1
Valves per cylinder	4
Max boost pressure	2.5 bar (abs)
Max engine speed	8000 rpm
Max power	> 300 hp

The behavior of the engine and of the turbocharger in the region below and above the critical rpm at which the air restrictor is choking (rpm*) is very different. This aspect is to be highly taken into account considering the working characteristic of the engine and its application. For example the maximum torque can only be reached before the rpm*, where the turbocharger can be exploited, although the maximum power can be reached beyond the rpm*, where the engine is no more working in full load. Typically, the target of a race engine is to achieve the maximum power and efficiency at higher rpm, not compromising too much the drivability and torque at lower rpm. In order to do this a proper design of the intake/exhaust system, as well as of the combustion chamber and of the injection pattern is mandatory.

In this kind of engine a high compression ratio (>10.5) is mandatory for a good performance, thus knocking is the main limiting issue for power output. To keep it under control a robust and fast combustion over the rpm has to be obtained. In the definition phase of the intake channel design and of the injection pattern, particular attention has been paid on mixture formation and turbulence level at ignition point (IP), as well as flame propagation by means of detailed 3D-CFD analyses.

According to previous experience on similar boosted engines and the knowledge gained during the project, the diagram on Figure 1 shows a qualitative trend over rpm of turbulence level and mixture quality, which takes into account the presence of high inhomogeneity or residual gas in the chamber. These two parameters are, according to the authors, the best indicators for robust and fast flame propagation. It can be observed how the knock tendency reaches its maximum over a wide range of rpm, and sinks only beyond rpm* to a non-critical level. This trend is the most responsible for achieving the maximum power beyond rpm*, as shown in Figure 2.

Only before rpm* the maximum boost pressure of 2.5 bar absolute can be reached and there the high cylinder filling is responsible for the high knock tendency (Figure 1). Moreover, in combination with other factors, as the engine is DI fueled, the mixing process plays a fundamental role in the combustion duration determination. As turbulence is nearly proportional to rpm, at low engine speeds the mixture quality can be insufficient to guarantee a fast combustion and thus to reduce knock tendency.

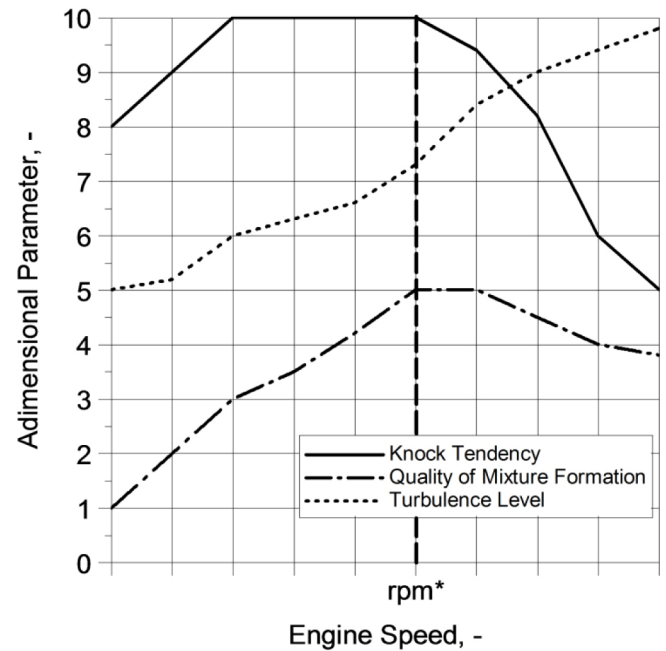


Figure 1. Qualitative trends of knock tendency, turbulence and mixture quality in the cylinder (WOT)

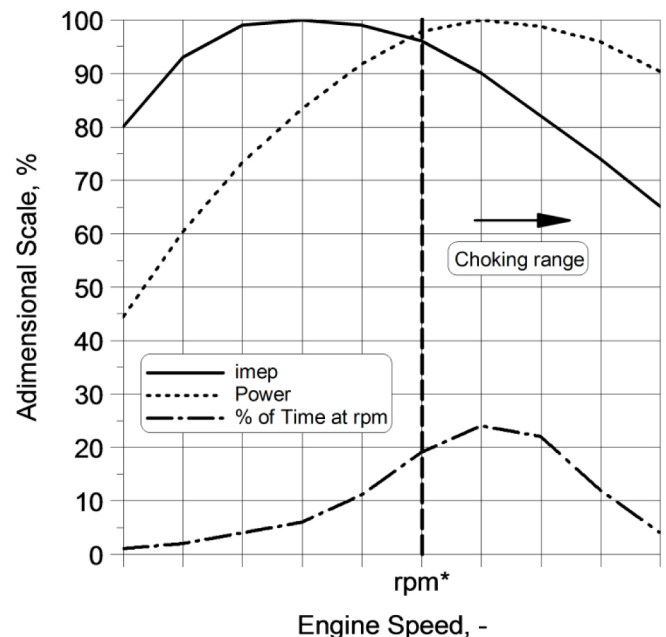


Figure 2. Trends of imep, power (at WOT) and example of expected percentage of time spend on a certain engine speed

On basis of the observed diagrams the target operating range of the engine was defined. A proper fluid change strategy, valve timing and the intake/exhaust design layout was investigated by means of a fast reliable 1D-CFD tool, although at the early stage of the development only rough combustion rate profiles were available from similar engines.

and only a rough estimation about the IP (maximum pressure, knocking) could be made by means of GT-Power (Gamma Technologies) and of the cylinder module *FKFS-User Cylinder* [1] for a better estimation of the combustion. Once the 1D-CFD model was built, the main design features were optimized, e.g. valve timing, intake runners and exhaust manifold design. The use of the 1D-CFD model was helpful also to understand the behavior and working points of the turbocharger (imposed by the regulations and supplied by Garrett) in particular in the vicinity of the choking speed (rpm^*), also when the first measurements from engine prototypes became available.

The 1D-CFD model allowed a rapid definition of many parameters and design features, but its limit was represented by the impossibility to predict optimal intake channel shape or the effect of a different combustion chamber as well as a different injector pattern. Only by means of the more accurate 3D-CFD analysis this could be possible. The boundary conditions of the 3D-CFD were supplied by the GT-Power model since no test bench data were available for a long period, and the intake/exhaust designs were changing quite rapidly during the first project phases.

3D-CFD ANALYSES

Quicksim

The 3D simulation tool used to perform the engine analysis is called “*QuickSim*” and its aim is to analyze the same systems simulated in the 1D-CFD or measured at the test bench: from a single-cylinder analysis, e.g. for combustion process development, up to the full engine for global investigation and performance development as required in motorsports.

The main peculiarities of this 3D-CFD tool are:

- Tool dedicated and optimized for the simulation of internal combustion engines.
- Fast response thanks to improved, engine specific CFD models able to ensure reliable results also using coarse meshes: the whole mesh presented in [Figure 3](#) is constituted by about 500,000 computational cells (only 40,000 cells are required for the combustion chamber discretization).
- Ability to simulate a full engine configuration for many consecutive operating cycles, as the CPU time required to simulate one operating cycle with one CPU is approximately equal to 14h.
- No limitations concerning the engine design, fuel injection or valve motion.
- Clear and comprehensive analysis of the engineering-scientific relevant engine parameters with tables, diagrams, images or animations.
- High flexibility given by the possibility to fulfill the technical, temporal and financial requests of the project.

In the following chapter some more details dealing with the injector model implemented in the code will be provided.

For a detailed description of the *QuickSim* code the reader can refer to [2].

Tumble vs. Filling

One of the most difficult challenges during the design of an engine fitted with an air restrictor is to find the best trade-off between cylinder filling and turbulence intensity (that highly influences the combustion duration); that because before rpm^* the air restrictor allows a sufficient air mass flow to fully fill the cylinders, but on the other side after rpm^* the air mass flow entering the cylinders can only decrease even with the best intake channel shape. As well known from bibliography [3,4] an intake channel that allows a good filling does not provide a sufficient turbulence (especially at lower engine speed) and vice versa.

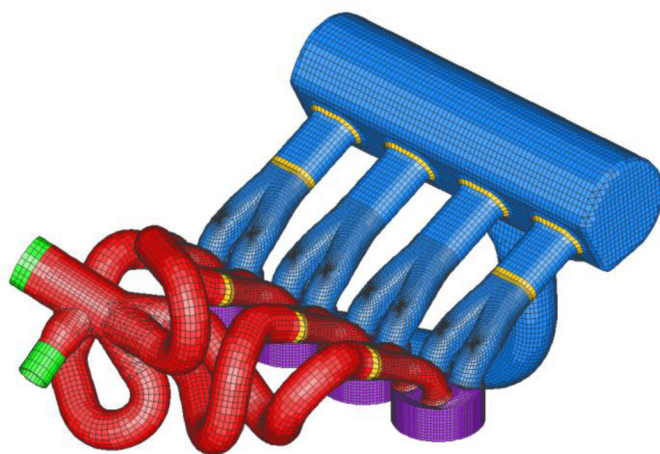


Figure 3. 3D-CFD-domain of one development variant of the full engine

During the described engine development process, the 3D-CFD *QuickSim* code has been employed to optimize the intake channel design, finding the best compromise to reach a good driveability for $\text{rpm} < \text{rpm}^*$ and the best performance for $\text{rpm} > \text{rpm}^*$.

Due to confidentiality reasons, these investigations and their details can not be presented; it will be possible instead to provide the details of the analyses dealing with the injector modeling.

INJECTION MODELING

The numerical modeling of the injection process is of the utmost importance in order to realize an accurate prediction of the mixture formation and therefore of the combustion process, but it still poses some questions. In the following some particular aspects of the injection modeling dealing with gasoline multi-holes, spray targeted injectors will be discussed. In order to realize these analyses at the early stage of the engine development, the research institute *VW Forschung* (Wolfsburg) has realized and made available some experimental data in a constant volume spray chamber with optical access; the relative experimental conditions are

summarized in Table 2. The data presented here are relative to the spray generated by four variants of the same injector. All these four variants have five holes and the same directions for the single jets plumes; they differ only as they can realize two different mass flow ratios, and for each mass flow ratio there are two variants realized with two different technologies: the classical spark erosion one and the new laser drilling technology. The injector variants will be identified with a conventional acronym described in Table 3: the first character is representative of the drilling technology and the second character distinguishes the two different values of mass flow rate tested.

Table 2. Experimental conditions of the spray chamber

Chamber pressure [bar]	1.6
Chamber temperature [°C]	100
Injection temperature [°C]	90
Rail pressure [bar]	200
Injection duration [ms]	1.0

First of all, an experimental datum that results highly valuable in the numerical simulation setting is the measured mass flow rate. The curves relative to the injector variants LH and EH are presented in Figure 4 and it clearly appears how the availability of this kind of data for the injection model set-up allows a much more reliable injector simulation with respect to a presumed mass flow rate curve.

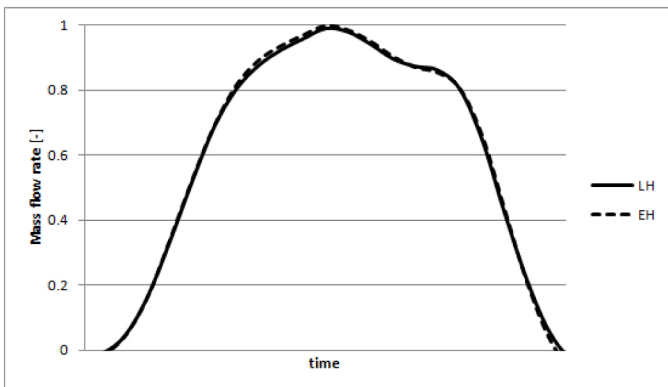


Figure 4. Mass flow rate realized by the injector variants LH and EH

The points of view from where the images have been taken, identified as “front” and “side” view, are indicated in the schematic presented in Figure 5. Specifically, in the experiments, the injector has been aligned by means of its electrical connector according to the presented schematic that shows the reference coordinate system provided by the injector manufacturer. Thus, the rotation angle with respect to the ideal configuration for the visualization (that is the one that provides symmetrical “side” images) is exactly equal to 25°.

Table 3. Injector definition

		Drilling technology	
		Laser	Erosion
Mass flow rate	High	LH	EH
	Low	LL	EL

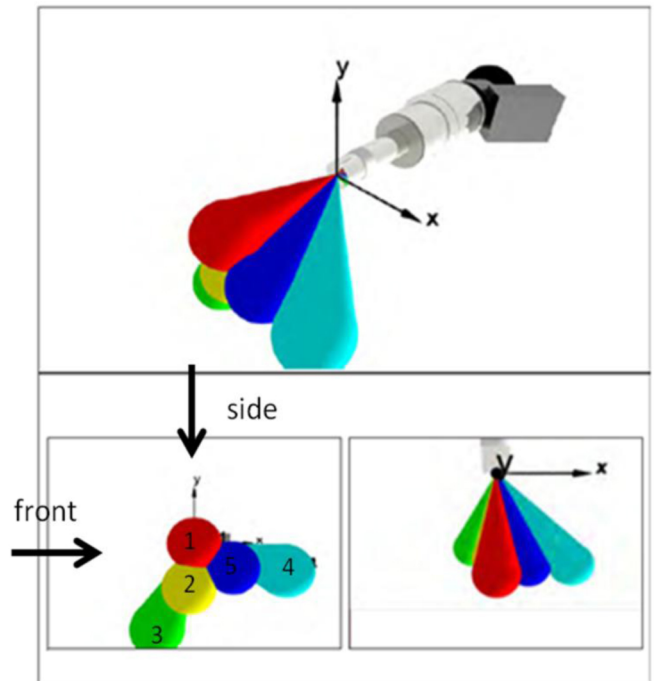


Figure 5. Injector geometry configuration, nozzle numbering and location of the image views

In Figure 6 the jet patterns of the spray generated by the four injector variants under investigation are presented; the axes of the spray plumes from each nozzle are marked with the corresponding color from the schematic in Figure 5. As one can see, even if the geometrical directions declared by the manufacturer are the same for all the injector variants, the orientations of the five jets result very different. In particular, the jets generated by the injectors realized with the same technology, but with a lower mass flow rate, show a “wider” shape; this means that each spray plume is at a higher distance with respect to the injector axis. This behavior has been identified even for other injectors.

Even the drilling technology employed to realize the injector seems to noticeably affect the spray pattern: the laser drilled injector variant, being equal the mass flow rate, generates wider spray plumes, both in terms of single beam angle and in terms of jets orientation, that show higher distances from the injector main axis. The first behavior has also been noticed by Jochman et al. [5] for a six - holes injector. The authors gave even a convincing explanation for this phenomenon: the inlet holes edges realized with the laser