

Fluid Dynamic Optimization of a Moto3™ Engine by Means of 1D and 1D-3D Simulations

Gianluca Montenegro, Tarcisio Cerri, Augusto Della Torre, and Angelo Onorati
Politecnico di Milano

Marco Fiocco and Davide Borghesi
Mahindra Racing Spa

ABSTRACT

In this work an integration between a 1D code (Gasdyn) with a CFD code (OpenFOAM®) has been applied to improve the performance of a Moto3™ engine. The four-stroke, single cylinder S.I. engine was modeled, in order to predict the wave motion in the intake and exhaust systems and study how it affects the cylinder gas exchange process. The engine considered was characterized by having an air induction system with integrated filter cartridge, air-box and intake runner, resulting in a complex air-path from the intake mouth to the intake valves, which presents critical aspects when a 1D modeling is addressed. This paper presents a combined and integrated simulation, in which the intake systems was modeled as a 3D geometry whereas the exhaust system, which presented a simpler geometry, was modeled by means of a 1D approach. Predicted 1D and 1D-3D results, concerning pressure pulses in the intake system and volumetric efficiency, were used to guide the design stage in order to improve the engine performances in terms of torque and power output. Calculation carried out on prototyped geometries have been successfully compared to experimental measurements.

CITATION: Montenegro, G., Cerri, T., Della Torre, A., Onorati, A. et al., "Fluid Dynamic Optimization of a Moto3™ Engine by Means of 1D and 1D-3D Simulations," *SAE Int. J. Engines* 9(1):2016, doi:10.4271/2016-01-0570.

INTRODUCTION

In the last decade internal combustion engines have experienced a phase of deep and intense investigation finalized mainly to the optimization of performance and fuel consumption. In the field of high performance engines, both for automotive and motorbike applications, the attempt to reach higher and higher levels of power output has made the design process one of the most critical step in the engine production process. A high performance engine requires a rather carefully tuned intake and exhaust system, both characterized by low pressure drops, in order to reach the maximum power output that the engine can potentially offer. To accomplish the different requirements imposed by the emission legislation and by the market, an important effort was carried out by scientists and researchers for the optimization of the combustion process and for the design of the intake and exhaust systems. Hence, these last components are becoming of primary importance not only for the role they play on the engine volumetric efficiency, but also for their influence on the abatement of the radiated noise and on the catalyst light off time. In this scenario fluid dynamic simulation tools have encountered an increased demand to assist the design phase, mainly because of their flexibility and accuracy. As a matter of fact, the prototyping procedure could become sensibly expensive and time consuming, if many configurations have to be tested. The aid of CFD tools can remarkably reduce the duration and the costs of this stage. However, the numerical simulation, if not correctly handled, can produce the opposite effect. For instance, a fully multidimensional analysis of a

multi cylinder engine coupled to its own intake and exhaust system might become a challenge even using a PC cluster. A factor that has to be considered is the possibility of relying on simpler and lighter calculation tools. Large systems, in fact, are often analyzed using one dimensional simulation tools, to save time and simultaneously preserve computational resources. Definitely, 1D thermo-fluid dynamic models are the most common tools in the automotive industry for engine design, due to the low computational effort required and the straightforward application by the users. Moreover, 1D models guarantee a high accuracy in representing the whole engine coupled to its intake and exhaust systems, for any speed and load condition [1, 2, 3, 4]. However, the intrinsic limit of 1D simulation is related to the fundamental assumption of planar wave motion, which is not realistic in complex 3D shapes of typical components of the air management system (plenum, manifolds, silencers, etc.). For some applications a hybrid 1D-3D simulation approach, based on the coupling of the two codes, can be a useful technique: it allows to keep the simplicity of the 1D approach for most of the calculation domain and to reserve the complexity of the 3D approach to a minimum of calculation domains, for which the 3D modeling is clearly more suitable. For example, complex shape components, such as air boxes with filter cartridge, intake and exhaust manifolds, silencers, multi-pipe junctions, etc., are generally characterized by a significant non-planar wave motion, so that the 3D approach is definitely more correct [5]. In this work a coupled 1D-3D code has been applied to the simulation of a high performance single

cylinder, SI engine for a Moto3™ motorbike. The 1D code used is GASDYN, a code developed by the authors during the last two decades [6] for one-dimensional modeling of IC engines. The 3D CFD code is OpenFOAM coupled to a library, the Lib-ICE, specifically developed for internal combustion engine simulation [7, 8]. The coupling numerical procedure is based on a Godunov type Riemann solver, implemented both in the 1D and in the multi-D code, to ensure stability and robustness of the method [9, 10]. The two codes have been coupled by applying the same Riemann solver at the interface between the 1D and the 3D domains. In this way, during the simulation the information is continuously passed back and forth between the two codes, resulting in a complete integration time step by time step. The hybrid 1D-3D code has been used to investigate the influence of different layouts of the intake system (air-box and intake runners) on the engine performance as function of the rotational speed. The most promising configuration is finally optimized by means of both 3D simulation and experimental measurements on the test bench.



Figure 1. Picture of the Mahindra MGP30 bike, which runs in the Moto3™ 2015 World Championship.

ENGINE-VEHICLE COUPLING

The shape of the engine performance curve is fundamental to study the coupling between the engine and the vehicle (in this particular case a racing motorbike). This type of vehicle needs the maximum performance bound within a limited range of revolution speeds, due to the use of a close set of gear ratios, which constraints the usage of the power unit, at wide open throttle, towards high revolution speeds. Ideally, to better exploit the available power, the engine should provide, at the wheel, constant power at every vehicle speed:

$$P_e = (F_m)_{opt} \cdot v = \text{constant} \quad (1)$$

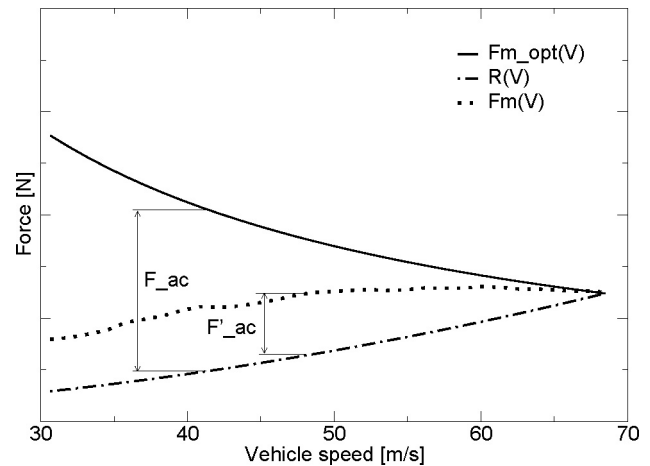


Figure 2. Chart of the available driving force at the wheel (engine characteristic) and of the drag force (vehicle characteristic) as a function of vehicle speed.

where $(F_m)_{opt}$ is the optimal driving force at the wheel, v is the vehicle velocity, P_e is the power at the wheel. In this case the driving force increases, with a hyperbolic law, as a function of the vehicle speed (as shown in Figure 2). The drag force $R(v)$, which is opposed to the vehicle movement, is a function of the speed, typically expressed as a sum of two contributions:

$$R(v) = R_0 + R_2 v^2 \quad (2)$$

where R_0 does not depend by the speed (it considers the slope of the track, e.g.), $R_2 v^2$ is dependent by the speed since it considers the aerodynamic drag and the friction of the tires. In Figure 2 the intersection between the $R(v)$ and $F_m(v)$ represents a stable operating condition, whereas the curve $(F_m)_{opt}$ is the optimal driving force. For operating points on the left of this intersection, the difference $F_m - R$ represents the maximum force available to accelerate the vehicle. Usually, the characteristic curve $F_m(v)$ of an internal combustion engine (that is similar to its torque curve), is different from the optimal driving force $(F_m)_{opt}$. This fact could determine that the accelerating force F'_{ac} available at a certain velocity is lower than the optimal one (F'_{ac}). In this case, the vehicle response, when a gear change occurs, becomes slow if compared to the ideal case. For racing vehicles, the particular gear box, allows to better approximate the optimal driving force, because of the higher number of gear ratios and because of the torque curve of the engine, which, at high engine speeds, has a slope close to the hyperbolic curve.

ENGINE SPECIFICATIONS

The engine taken into account in this study is a Moto3™ engine, whose main specifications are summarized in Table 1. For what concerns the performance of this engine, Figure 3 reports the curves of the accelerating force (at WOT operating point) relative to each gear ratio, for a specific race setup of the Moto3™ world championship. It is evident that the 6th gear ratio allows achieving the

maximum vehicle speed in proximity of the maximum engine revolution speed, whereas the ratios below the 6th gear appear to be suitably spaced in order to have the optimal accelerating force at the speed when the gear is changed. In this way, the optimal driving force ($F_{m,opt}$) is well approximated by the curves of the accelerating force at high engine revolution speed.

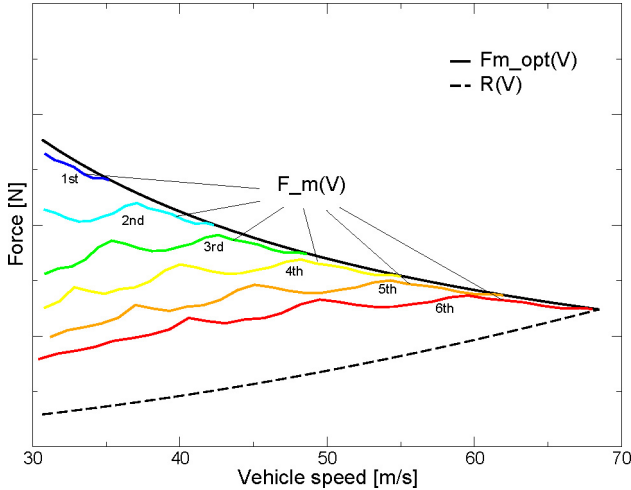


Figure 3. Chart of the available driving force at the wheel parameterized to the gear ratios (engine characteristics) and of the drag force (vehicle characteristic), as a function of vehicle speed. The data are referred to the studied engine for a specific race setup of the Moto3™ world championship.

Table 1. Technical specifications of the Moto3™ engine.

Maximum swept volume	0.25 dm ³
Maximum rev. speed	13500 rpm
Air metering	Naturally aspirated
Cylinder arrangement	Single cylinder
Maximum cylinder bore	81 mm
Fuel supplier	PFI
Max. injection pressure	0.5 MPa
Engines per season	6 unit/driver
Refueling	Forbidden

GOVERNING EQUATIONS

1D Model

The 1D model adopted for the simulation is a research code developed by the authors (GASDYN) [11, 12]. In the 1D model, the conservation equations of mass, momentum and energy are expressed in a strong conservative form and applied to the simulation of complex systems of pipes, with source terms to account for the friction (f_w) and the heat transfer (q) between the gas and the walls:

$$\frac{\partial (\rho S)}{\partial t} + \frac{\partial (\rho u S)}{\partial x} = 0 \quad (3)$$

$$\frac{\partial (\rho u S)}{\partial t} + \frac{\partial (\rho u^2 S + p S)}{\partial x} = p \frac{\partial S}{\partial x} - \frac{2}{D} \rho u |u| f_w S \quad (4)$$

$$\frac{\partial (\rho e_0 S)}{\partial t} + \frac{\partial (\rho e_0 u S + u p S)}{\partial x} = \rho (q + q_{re}) S, \quad (5)$$

where S represents the pipe cross section. The system of equations is closed with the perfect gas equation of state. The set of equations is then solved by applying explicit Godunov type methods based on the HLLC Riemann solver and on its extension to the second order accuracy (WAF) [13]:

$$W_i^{n+1} = W_i^n + \frac{\Delta t}{\Delta x} (F_{i-\frac{1}{2}} - F_{i+\frac{1}{2}}) - \Delta t C_i^n \quad (6)$$

where the inter-cell fluxes $F_{i\pm\frac{1}{2}}$ can be evaluated according to a first order approach or to its second order formulation [13]. Moreover, the source term C is discretized according to the stencil proposed by other authors [14].

Multi-Dimensional Model

The OpenFOAM code was adopted for the multidimensional simulation [15, 16]. It is a general purpose finite volume solver, written in C++ object oriented programming language, and provides the user with suitable libraries to discretize partial derivatives operators applied to arbitrarily oriented meshes. In this context it has been used to solve the full set of Navier Stokes equations describing a compressible and viscous unsteady flows. Due to the large size of the intake system modeled in 3D, the viscous losses occurring in the system cannot be neglected anymore, if a precise evaluation of the impact that a certain configuration has on the engine volumetric efficiency has to be evaluated. Additionally, since the effects of local cooling caused by the evaporation of injected fuel will be taken into account, also thermal diffusion effects will be considered in the equations, leading to the adoption of a full set of Navier Stokes equations.

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{U}) = 0 \quad (7)$$

$$\frac{\partial (\rho \vec{U})}{\partial t} + \vec{\nabla} \cdot (\rho \vec{U} \vec{U} + p \vec{I} - \vec{\tau}) = \rho \vec{f}_e \quad (8)$$

$$\frac{\partial \rho E}{\partial t} + \vec{\nabla} \cdot (\rho H \vec{U} - k \vec{\nabla} T - \vec{\tau} \cdot \vec{U}) = \rho \vec{f}_e \cdot \vec{U} + q_H \quad (9)$$

where $\vec{\tau}$ is the shear stress tensor, $\vec{f}_e$ is a volume force used to mimic the flow resistance of a generic porous media (for instance a filter cartridge), and q_H is the heat flux source term introduced to mimic the effect of fuel evaporation. The set of equations is then closed by the perfect gas equation of state and by the standard k-ε turbulence model transport equations.

1D-3D Coupling

A good compromise between accuracy of the prediction and simulation run-time is the adoption of integrated 1D and 3D simulation tools [17]. This strategy is also adopted by other commercial 1D software, which have been coupled to CFD codes to allow the detailed study of wave propagation inside of complex geometries. Many works concerning this topic have also been published during the last years [18, 19, 20]. To take into account the effects of eventual variation of the multidimensional domain on the 1D side, the so called “strict coupling” has been proposed. In this procedure, the two codes are passing the information forth and back at each time step in such a way that one domain is affected by what happens in the other one. In order to truncate the 3D domain in regions close to highly 3D shapes, an advanced procedure has been adopted, therefore allowing the treatment of non-uniform fields on the 3D side without compromising the stability of the solution. This approach involves the solution of the Riemann problem for every cell which owns the face constituting the boundary surface and is fully described in recent papers [9]. Another important reason why the coupling between the 1D and the CFD models has been used is the advantage to carry out simulations involving the engine as a pressure pulsations source. This condition at the boundary cannot be easily reproduced resorting to a boundary condition approach. As a matter of fact, imposing the pressure trace, as a function of the time, or of the crank angle, could not be enough to reproduce the exact flow condition occurring in the exhaust, or in the intake system, since a mismatch between the measured and calculated mass flow may occur. In this work only the intake system has been modeled as 3D. In particular the 3D domain extends from the intake mouth to 3 cm downstream of the intake runner open end, right after the trumpet shaped pipe. Therefore the remaining portion of the intake runner has been modeled as 1D.

1D MODEL CALIBRATION

Characteristic Lengths

As can be seen from Figure 4, the air box geometry is composed by a suitably shaped plenum (in order to adapt to the motorbike components and to the pilot position) and it is connected to the ambient and to the cylinder by an intake pipe and intake runner, respectively. This last component is equipped with a trumpet shape section extended into the plenum to limit the pressure loss of inflow. A simplified way to model the intake system would involve the usage of 0D elements, such as 0D volumes, whose main function will be to mimic the filling and emptying behavior of the plenum, whereas the characteristic length of the whole system will be given by the length of the inlet pipe. This kind of space discretization is not convenient when a detailed analysis of the wave action is addressed, mainly because the capturing of the acoustic length is misled by the 0D assumption. It is suitable when there is an interest to analyze only cycle averaged quantities. Therefore, the approach followed in this work is based on the reconstruction of volumes with equivalent 1D pipes. An example of how the 1D schematic has been built to capture

all the acoustic length is given in Figure 5. It can be noticed that the steering rod seat, visible in Figure 4, has not been modeled as a flow split into two branches, with a following joining of the two pipes. Instead, it has been modeled by a pressure loss boundary condition, that takes into account the split into two streams and the following flow joining downstream with partial pressure recovery [14]. The main volume is reduced to a single pipe whose length is given by the distance between the filter cartridge and the trumpet axis, in order to capture the equivalent wave path, and whose diameter has been set to match the volume of the plenum. The two column resonators, identified by the two pipes with a closed end on one side, are used to capture the transversal wave motion originated by the reflections of pressure pulses coming from both the intake runner (as an effect of the pipe extension inside the volume) and the intake pipe. The intake runner is therefore modelled as a pipe connected to the main volume by means of variable flow section to reproduce the trumpet shaped pipe. According to the acoustic theory [14, 21], the open end of a pipe extended into a volume needs to be corrected, in order to capture the effects of non planar wave generated inside the volume. This correction usually results in a length correction equal to a fraction of the pipe diameter (as shown in Figure 7). In this particular case, the end correction that has been used is $\delta = 0.3D$. As previously said, this type of discretization permits to capture both dynamic effects of wave actions and of filling and emptying behavior. Similar consideration can be done for the exhaust system, where, for this particular case, the pipe system is easily reduced to a 1D model, thanks to its simplicity. Once the fluid-dynamic schematic has been correctly done, the calibration of the combustion model was addressed. In order to predict realistic values for the heat release law at each engine speed, and consequently achieve a good prediction of the brake torque, a multi-zone, quasi-dimensional combustion model has been used, to evaluate the thermodynamic and chemical gas properties inside the combustion chamber [22]. The calculation of the fully developed turbulent flame speed has been performed resorting to the Zimont model [23]. Despite the adoption of the quasi-dimensional approach, to account for the turbulence generation and dissipation, the turbulent intensity u' has been taken into account and its value has been determined by the two equation model k-K [22]. The Woschni model [24] was adopted to calculate the combustion chamber wall heat flux. The resulting curve for the brake torque of the engine is shown in Figure 6, where it can be noted an overall good agreement between measured and calculated values. However, the peak torque seems to be predicted with a slight shift (around 250 rpm) with respect to the revolution speed. It appears as if the 1D model were tuned towards higher revolution speeds. This may be due to principally two effects: gas temperature and pipe length, since they are the two parameter affecting the propagation time of pressure waves. Looking at the fuel injection system, it is likely that the amount of fuel injected may cause, as a consequence of the evaporation, a local decrease of gas temperature, which brings to a slower pressure wave propagation. Therefore a 3D analysis must be performed in order to evaluate the impact of fuel vaporization.