

Guidelines for the Optimization of a Muffler in a Small Two Stroke Engine

Francesco Testa
Universita degli Studi di Modena

Vincenzo Gagliardi and Marco Ferrari
EMAK Spa

Stefano Fontanesi
Universita degli Studi di Modena

Andrea Bertani
EMAK Spa

ABSTRACT

It is well known that 3D CFD simulations can give detailed information about fluid and flow properties in complex 3D domains while 1D CFD simulation can provide important information at a system level, i.e. about the performance of the entire engine. The drawbacks of the two simulation methods are that the former requires high computational cost while the latter is not able to capture complex local 3D features of the flow. Therefore, the two simulation methods are to be seen as complementary, indeed a coupling of the two approaches can benefit from the pros of the two methods while minimizing the cons. In particular, with a multi-scale modeling approach (1D-3D) it is possible to simulate large and complex domains by modeling the complex part with a 3D approach and the rest of the domain with a 1D approach.

This paper describes an optimization cycle analysis of the unsteady flow of a single cylinder, two stroke gasoline engine using advanced numerical tools, which are in turn validated by means of experimental measurements. In particular, a 3D model (based on STAR-CD code) of the entire engine and a 1D-3D integrated fluid dynamics model (based on GT-POWER 1D and Converge-LITE 3D codes) is developed and applied for the representation of the geometrical domain and for the prediction of performance and gas-dynamics in the whole intake and exhaust systems.

The methodology allows users to accurately predict and deeply understand unsteady phenomena in the whole engine and capture the wave motion, which strongly affects the muffler 3D design in small two stroke engines equipped with resonance pipes.

CITATION: Testa, F., Gagliardi, V., Ferrari, M., Fontanesi, S. et al., "Guidelines for the Optimization of a Muffler in a Small Two Stroke Engine," *SAE Int. J. Engines* 9(4):2016, doi:10.4271/2016-32-0050.

INTRODUCTION

Two-stroke internal combustion engines are usually preferred to four-stroke engines when less complex and cheaper solutions are needed [1]. Furthermore, two-stroke engines are still preferable in some applications for which low maintenance costs and manageability, due to the small weight/power ratio, are indispensable; examples are small portable tools for gardening or agriculture (chainsaws, brush cutters, gas saws). For such engines power and torque are surely the most important design targets; the reduction of unburned hydrocarbon (UHC) emissions and fuel consumption still remains an important issue.

To reach the mentioned targets without penalties in the emissions it is often necessary provide the muffler with a catalyst; at the same time, a lot of efforts are concentrated to limit UHC production before the catalyst, in order to comply with the regulations even after a high number of working hours [2, 3].

As it often happens for the two-stroke engines, it is possible to improve engine overall performance with a suitable design of the exhaust system thanks to the exploitation of pressure waves generated during the scavenging process, as shown in previous work carried out by Gustafsson, Blair and Jonsson in 2001 [4], 2006 [5] and by Zsiga, Kerres, Bach and Fouss in 2010 [6]. In this case a resonant pipe is placed inside the muffler in between the exhaust port and the outlet orifice. Since small variations of lengths and diameters lead to

significant changes in engine-out performance, a long phase of one-dimensional studies is usually needed to tune and optimize the engine. At the same time, the designer needs to comply with limitations in the overall dimensions of the machine. As a consequence, the geometry of the resonant tube usually deviates from a predominantly axisymmetric geometry and the 1D model fails to capture complex 3D fluid dynamics features taking place inside the system.

In this paper a methodology based on coupled 1D-3D fluid dynamic tools is proposed and applied to a 2-stroke engine to provide a robust instrument able to accurately estimate the effects of design changes on power and fuel trapping. All assumption and settings are validated against experiments on a next-generation commercial hand-held gas saw machine. The validity of the methodology in predicting the most relevant phenomena affecting engine performance is finally demonstrated. The procedure followed is summarized in [Figure 1](#) below.

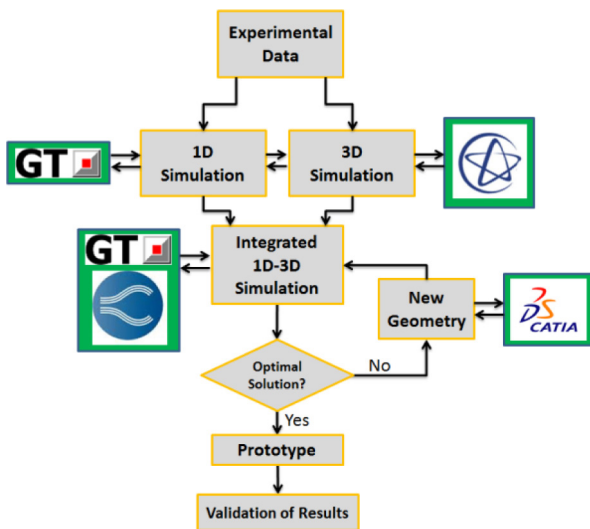


Figure 1. Flow chart of the overall project

Gas-saw Machine Layout

The handheld machine considered in the present paper is a gas saw unit designed by EMAK ([Figure 2](#)); it is equipped with a two-stroke engine having the characteristics listed in [Table 1](#).

Table 1. Engine overall dimension

Displaced volume	63.3 cc
Stroke	35 mm
Bore	48 mm
Connecting Rod	58 mm
Compression ratio	10.6:1
Exhaust Port Open	102° ATDC
Transfer port Open	130° ATDC
Inlet port Open	71° BTDC



Figure 2. Handheld machine considered in the present paper

The 2-stroke engine is a traditional unit with 4 transfer ducts; the mixture is provided by a butterfly valve diaphragm carburetor with a diameter of 13.5 [mm]. Concerning the muffler, the initial design is replaced by a modified one composed of two half-shells which enclose a resonance pipe as shown in [Figure 3](#): a wire-mesh converter catalyst is placed before the final orifice; the catalyst is a coated 40 × 12 mm device whose coating is based on a mixture of precious metals: platinum (Pt), palladium (Pd) and rhodium (Rh) with the ratio: 4: 10: 1 (pt: Pd: Rh).

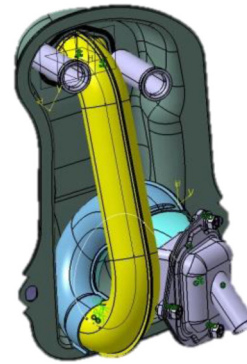


Figure 3. Muffler with resonance pipe

EXPERIMENTAL SETUP

The engine is initially tested on a dynamometer test bench in order to monitor the standard performance parameters (Power, Torque, Delivery Ratio, Air-to-Fuel-Ratio (AFR)) and to check if emissions comply with the limits imposed by regulations. Such preliminary data set is also used for simulation calibration purposes. For the performance optimization stage the engine is equipped with a pair of quartz pressure transducers: one (AVL GH14D) placed in the combustion chamber while the other (KISTLER 6052CS31U40) in the crankcase; a magnetic angle encoder is installed on the shaft, allowing to accurately phase the combustion pressure. In-cylinder pressure and related parameters, such as the coefficient of variation of the indicated mean effective pressure, the burning gravity center and the rate of chemical energy release, are evaluated on an individual cycle basis and then averaged over 100 cycles as shown in [Figures 4](#) and [5](#).

Since this category of engines is usually run with a rich AFR, Carbon Monoxide (CO) rate in the exhaust (CO %) is used as an index of engine operation conditions. All the engine parameters and data during the experiments are extra logged using the technique proposed

by R. Douglas [7], which allows to calculate two-stroke engine AFR based on exhaust gas emissions. The technique is also able to provide useful information on the scavenging process, such as delivery ratio, charging efficiency and trapping efficiency.

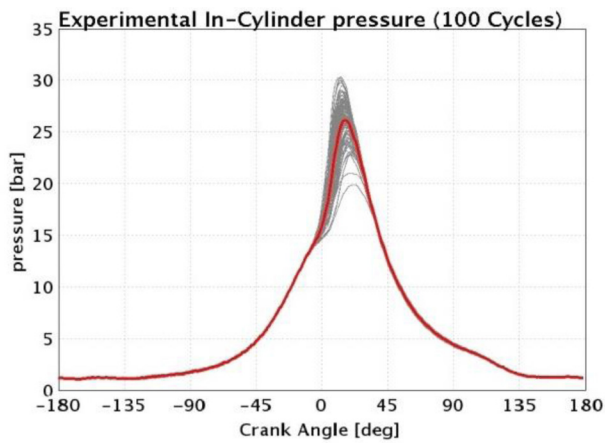


Figure 4. Experimental in-cylinder pressure trace

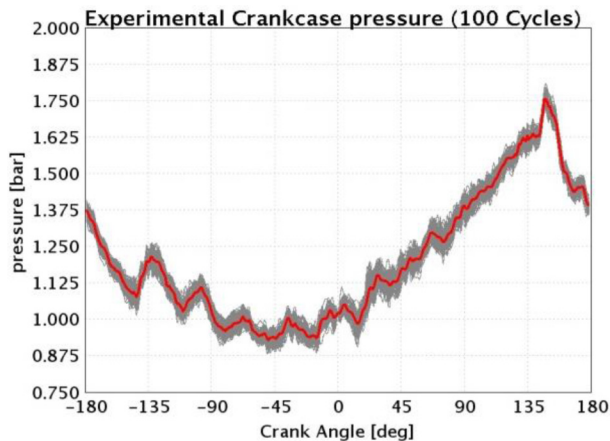


Figure 5. Experimental crankcase pressure trace

CFD METHODOLOGY

Gas motion can be modelled in 3D-CFD, e.g. solving the Navier-Stokes equations of gas motion numerically.

Such modelling needs accurate geometric information of the system, see e.g. the complex muffler geometry reported in Figure 3. The boundary conditions of the model, i.e. how the gas enters and leaves the modelled component, are a key factor for the reliability of the numerical model predictions.

Due to the complexity and the computational effort, 3D models are most often used to model only individual components or subsystems of the engine.

The obtained solutions provide valuable information on flow motion, which can be in turn used for reduced-complexity model families. The reverse is also true; reliable models of low-complexity structure can be used at a component level for detailed 3D simulations.

Nowadays, 0D and 1D codes are often used during the engine development stages to guide and limit expensive experimental analyses [8, 9, 10]. These codes solve conservation equations along pipes and in sub-volumes while they are generally based on empirical sub models [11, 12] to account for complex geometries and phenomena.

Their computational cost is reduced, if compared to 3D models, so that large parts of an engine system can be simulated with reasonably short simulation times.

The two main drawbacks of such tools are that a wide set of data is usually needed to better calibrate the engine model and that they lack of accuracy when complex 3D effects must be accounted for. In order to overcome this last limitation, a third simulation approach can be adopted: starting from a validated 1D model of the whole engine, an integrated 1D-3D approach may be used replacing part of the simplified 1D model with a 3D object, whose detailed representation is considered of primary importance for the overall accuracy of the numerical forecast. In the present paper, a choice is made to model the muffler as a 3D part. This kind of “coupled” approach is a good trade-off between the previously described methods in terms of time to solution and accuracy of the results.

In the present paper, such 1D-3D simulations are part of a comprehensive automated workflow aiming at the improvement of the muffler design. The workflow starts from the generation of different muffler geometries inside a CAD tool (here Catia V5) the different solutions are then passed to Converge Studio, a preprocessor software which allows to generate 3D fluid domains starting from solid bodies; to close the loop, the fluid domains are imported in the integrated 1D-3D simulation tool to evaluate the performance of the design solutions.

Concerning the operating point at which all the simulations are carried out, a choice is made for 8000 rpm WOT condition, which corresponds to the peak power operating point of the engine.

1D Simulations

The 1D model is generated starting from a CAD geometry of the whole machine assembly. The model (generated using GTPOWER Software) is comprehensive of all the engine components except the air filter.

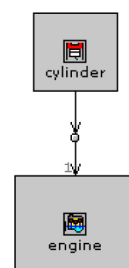


Figure 6. GT-Power 1D simplified model

As a preliminary step, a simplified model, shown in Figure 6, is generated to determine the burn rate (Figure 7) starting from incylinder pressure, scavenging efficiency and trapping efficiency. In

this separate procedure, the amount of fuel that is transferred from the unburned to the burned zone is iterated within each time step until the cylinder pressure matches the measured cylinder pressure. The resulting burn rate is then used as an input for the complete 1D model.

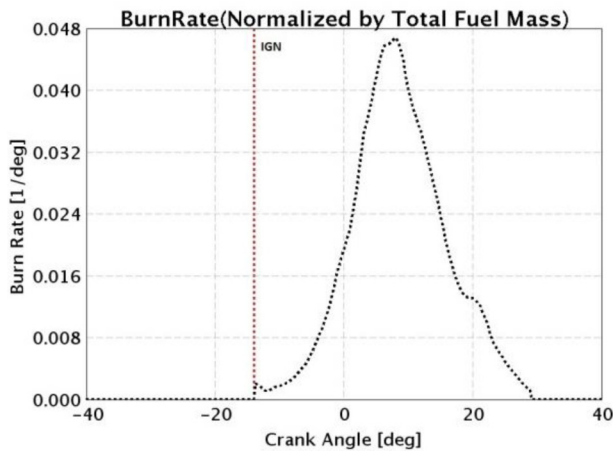


Figure 7. GT-Power 1D burn rate input 8000 rpm at WOT

This simulation requires engine geometry, cylinder wall temperatures, a heat transfer model, estimation of friction mean effective pressure and a proper set of both boundary and initial conditions. The geometry is extrapolated from CAD, while the other parameters are derived from a combination of experimental test data and user experience.

As an example, to estimate fmep (friction mean effective pressure) the pressure traces coming from experiments are used. In particular:

$$fmep = [imep - bmep - pmep] \quad (1)$$

Where the values for imep (indicated mean effective pressure) and pmep (pumping mean effective pressure) are related to in cylinder pressure trace and crankcase pressure trace respectively, while bmep (brake mean effective pressure) is related to the engine brake power.

After the model setup phase, a calibration of the 1D model is performed. In particular, a 3D model of the whole engine except the air filter is used, in order to validate the gas-dynamic behavior of the engine.

3D Simulations

To get a better agreement between experiments and 1D simulations, it is recommendable to generate a full 3D model to better calibrate the mass-fluxes at some relevant sections of the model (as an example, the discharge coefficients C_d extrapolated from the fully 3D model and used in the 1D model to calibrate the mass flux at the intake, transfer and exhaust port are reported in Figure 8).

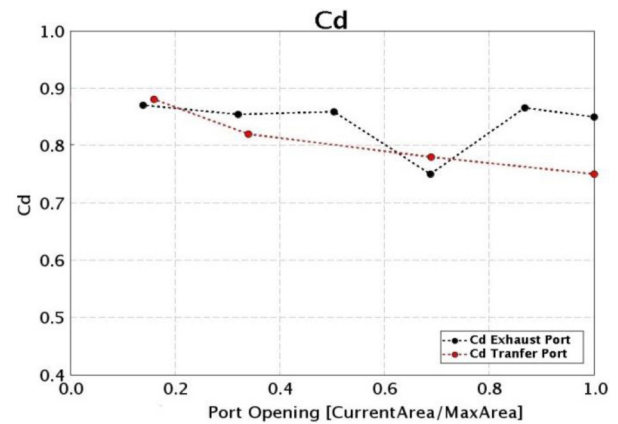


Figure 8. C_d derived from 3D CFD calculations

The full 3D model is generated starting from the solid CatiaV5 CAD of the engine and extracting the fluid domain reported in Figure 9. More in details, the model include:

- Combustion chamber;
- Cylinder;
- Muffler (including resonance pipe);
- Transfer ports;
- Exhaust port;
- Crankcase (including all piston features);
- Intake duct starting from carburetor

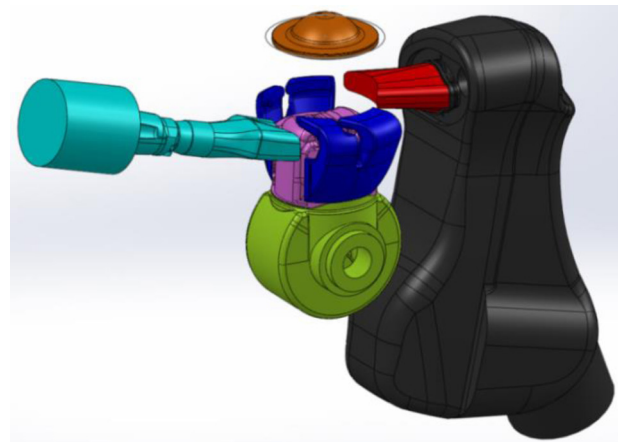


Figure 9. 3D CFD model domain

The generated model allows to account for the actual piston motion in both the head and the crankcase side; if necessary, the crankshaft rotation can be activated. Since the CFD approach is RANS based, the model runs until cycle to cycle convergence has been met. The crankcase region is simplified eliminating some minor geometrical details, but particular attention is paid to guarantee the correctness of the intake opening/closure time and the accurate representation of the inlet and transfer ports as well as to avoid mismatch in the crankcase compression ratio. The crankshaft and the connecting rod are modelled