

Development of a Low Pressure Direct Injection System for a Small 2S Engine. Part I - CFD Analysis of the Injection Process

Francesco Balduzzi, Giovanni Vichi, Luca Romani, and Giovanni Ferrara
University of Florence

Paolo Trassi, Jacopo Fiaschi, and Federico Tozzi
Betamotor S.p.A.

ABSTRACT

High specific fuel consumption and pollutant emissions are the main drawbacks of the small crankcase-scavenged two-stroke engine. The symmetrical port timing combined with a carburetor or an indirect injection system leads to a lower scavenging efficiency than a four-stroke engine and to the short-circuit of fresh air-fuel mixture. The use of fuel supply systems as the indirect injection and the carburetor is the standard solution for small two-stroke engine equipment, due to the necessity of reducing the complexity, weight, overall dimensions and costs.

This paper presents the results of a detailed study on the application of an innovative Low Pressure Direct Injection system (LPDI) on an existing 300 cm³ cylinder formerly equipped with a carburetor. The proposed solution is characterized by two injectors working at 5 bar of injection pressure. The injection nozzles are located in the liner wall at the opposite side of the exhaust port, with the axes oriented towards the piston at the BDC (Bottom Dead Center). The injectors are positioned above all of the cylinder ports in order to have an injection timing independent from the ports timing. The only constraint to the injection timing is related to the piston motion, with great benefits in terms of short-circuit reduction in all of the engine operating conditions.

Numerical and experimental activity was carried out in order to identify the best injector configuration and to assess the system performance. The results of the preliminary 3-D CFD study are here reported. The best positioning of the injectors was determined at the operating condition of maximum power by means of a sensitivity analysis of the circumferential position and spray orientation. The performance comparison was evaluated in terms of fuel short-circuit and spray vaporization. The optimized configuration was also verified at lower revolution speeds and partial loads showing a great potential for all of the working conditions. The experiments on a real prototype on the test bench confirmed the predictions showing significant improvements in comparison with the original configuration.

CITATION: Balduzzi, F., Vichi, G., Romani, L., Ferrara, G. et al., "Development of a Low Pressure Direct Injection System for a Small 2S Engine. Part I - CFD Analysis of the Injection Process," *SAE Int. J. Engines* 8(4):2015, doi:10.4271/2015-01-1727.

INTRODUCTION

The conventional small crankcase-scavenged two stroke engine is cheap, reliable and provides a high power-to-weight ratio. At the same time, it is affected by two critical problems: high brake specific fuel consumption (BSFC) and pollutant emissions, which include unburnt hydrocarbons and carbon monoxide. These issues are mainly related to the symmetrical port timing diagram combined with a carburetor or an indirect injection system. A part of the scavenge charge transferred from the crankcase, which is composed of a fresh fuel-air mixture, tends to short-circuit across the cylinder flowing directly through the exhaust port before its closing. This leads to an undesired loss of a high quantity of fuel.

The use of a direct injection strategy allows the introduction of the fuel separate from the scavenging air, minimizing the fuel short-circuit and substantially improving the potential of two stroke engines. By appropriately timing the injection, only air is lost, preventing the escape of unburned fuel from the exhaust port during the scavenging process. The whole amount of injected fuel is trapped in the cylinder and takes part in the combustion process, thus improving the engine efficiency and reducing the pollutant emissions.

Besides allowing a considerable reduction of the quantity of fuel short-circuited, direct injection leads to several secondary advantages: the thermodynamic efficiency could be improved due to the possibility of increasing the compression ratio; combustion with a stratified charge could also be performed, with the advantage of lower fuel consumption and heat losses. On the other hand, the complexity

of a direct injection system increases with respect to a carbureted two stroke engine. In the case of high pressure direct injection (HPDI) systems, high compression ratio pumps and high pressure injectors are needed, determining an increase also in the dimension, weight and cost of the system making this solution unsuitable for small 2S engines.

To overcome these drawbacks, several innovative technologies have been developed and introduced, with particular attention to LPDI solutions. The system designed by Ragg *et al.* [1] is based on a direct injection of fuel performed through an air-assisted method. In literature there are some examples of engines that operated with the working principle of Ragg's patent [2, 3, 4]. The fuel is injected into an annular chamber around the stem of a small poppet valve. The chamber is filled with compressed air. When opened, the poppet valve delivers the charge of fuel and air directly into the combustion chamber and the high relative velocity between the air and the fuel causes the atomization. The scavenge process is executed with air only. Similar to Ragg's patent, the solution proposed by Duret *et al.* [5,6] is designed with a pre-chamber in the cylinder head where the fuel is mixed with a small quantity of air derived from the crankcase, in order to obtain a rich mixture. The scavenge process is performed with air only, whereas the opening of a poppet valve lets the rich mixture enter into the combustion chamber when the ports are closed. The DICC system (Direct Injection Combustion Control) designed by Nino [7] utilizes a low pressure injector (5 bar) located above the transfer ports. The fuel is directly injected into the combustion chamber at the end of scavenging and the injection axis is tilted towards the engine head. The fuel atomization is due to the interaction with the fresh charge motion inside the cylinder, with the possibility of realizing a stratified charge. As an alternative, Winkler *et al.* from the Graz University of Technology [8] also developed a LPDI system (5 bar) with the injector placed into the cylinder liner, directly facing into the combustion chamber. Additionally, the presence of a casing opening in the piston skirt lets the spray flow into the crankcase when the piston is close to TDC. At high loads, the fuel is partially injected in the crankcase and the scavenging phase is performed with a very lean mixture, causing a minimum amount of short-circuit loss. All of the systems here reported guarantee a strong reduction of fuel consumption and pollutant emissions when compared to a common 2S engine.

The aim of the present study is to develop a low pressure direct injection system while at the same time preserving the simplicity of the design, the low weight and costs that are typical characteristics required of the small 2S engines. An existing 300 cm³ naturally aspirated two-stroke cross-scavenged carbureted engine, designed by Betamotor S.p.A., was converted into a LPDI engine. Two injectors working at 5 bar of injection pressure ensure the right quantity of fuel at both low and high loads. In a parallel study [9], the authors reported the results of an experimental activity where the performance of the engine in the LPDI configuration was evaluated at the test bench and then compared to the performance of the configurations provided with both the carburetor and the indirect injection system (PFI, Port Fuel Injection). The great benefits in terms of fuel consumption and pollutant emissions over the whole engine operating range are discussed.

The subject of this study was to perform a preliminary numerical analysis of the scavenge and spray processes of the LPDI system. In order to achieve an effective homogeneous air-fuel mixing, detailed three-dimensional simulations of the mixture formation must be carried out to assess the influence of the air fluid dynamics, injection system and strategy at different operating conditions. The best injectors' positioning was assessed by simulating the injection at different rotating speeds and loads in order to select the optimum configuration to be prototyped and tested [9]. The challenge in these simulations concerns the contemporary presence of a two-phase flow and complex geometries with moving boundaries. There are many numerical studies in literature that address the use of computational fluid dynamics (CFD) codes to model the in-cylinder fluid flow, turbulence and spray characteristics.

With particular attention to the simulations involving direct injection and fuel-air mixing, the Graz University of Technology presented different works [10, 11, 12] performed with commercial codes. They numerically and experimentally studied both low and high pressure direct injection strategies for two-stroke engines, also investigating the effects of the spray-wall interaction. Calculated and measured results were in a good agreement.

Concerning CFD investigations on low-pressure injections, the droplet and gas motion and fuel-air mixing is typically studied for PFI engines. Bianchi *et al.* [13] developed a numerical methodology able to predict the mixture formation for a four-stroke gasoline engine equipped with a three-hole PFI injector operating at a pressure of 250 kPa. Great effort was spent on obtaining a proper initialization of the secondary breakup model in order to achieve accurate predictions of spray characteristics. The investigation of the fuel-air mixture formation including the possible formation of liquid film was performed numerically by Lucchini *et al.* [14] with the open-source code OpenFOAM. The spray-wall interaction was first simulated in a pipe showing a rather good agreement with experimental data. Then the fuel air mixing process was simulated in a real SI engine geometry, where the formation of a tumble vortex was predicted and preserved during the compression process. Kim and Lee [15] focused on injector spray characteristics for a four-valve motorcycle gasoline engine. The experimental and numerical characterization of the spray behavior showed the great influence on the evaporation characteristics of the interaction with the air flow. Almost no film formation was observed, due to the high velocity of the flow.

ENGINE DESCRIPTION

The main objective of this study is the numerical simulation of the scavenging and injection processes of a 300 cm³ 2S engine cylinder characterized by a LPDI fuel supply system. The engine was developed from an existing naturally aspirated two stroke carbureted engine. The engine is loop-scavenged (Schnürle type) with five transfer ports and a single exhaust port. In particular, two lateral transfer ducts (T90) have a 90° orientation with respect to the engine symmetry plane, two ducts (T45) have a 45° orientation and the fifth transfer duct (T0) is opposed to the discharge. In [Table 1](#) the main engine characteristics are summarized.



Table 1. Betamotor 2S 300 cm³ engine data.

Type	Single Cylinder, 2-Stroke, Spark-Ignition, Liquid Cooled
Displaced volume	293.1 cm ³
Stroke	72 mm
Bore	72 mm
Connecting Rod	125 mm
Geometric Compression Ratio	12:1
Trapped Compression Ratio	8.2:1

Standard low cost components from the automotive market were used, without significant increase of weight. In particular, two injectors working at 5 bar of injection pressure were supplied by Synerject, ensuring the right quantity of fuel at both low and high loads. The main goal of the CFD analysis was the definition of the optimal positioning of the injectors. Since the fuel has to be injected directly into the combustion chamber, the injection nozzles were placed in the liner wall. This solution was preferred to the standard direct injection configuration, having a nozzle end typically disposed in the cylinder head, due to the lower thermal tolerance of the low pressure injector. Since positioned in the cylinder liner, the nozzle is surrounded by the water-cooling jacket and is protected by the piston during the combustion. In fact, in this phase the injector is covered without being directly exposed to the propagating flame. Therefore, the injection timing is independent from the port timing but is related to the piston motion, limiting the angular range at disposal for the injection.

NUMERICAL SETUP

A commercial code ANSYS Fluent [16] was used in a three-dimensional form to solve the time dependent unsteady Reynolds-averaged Navier-Stokes (U-RANS) equations. Turbulence closure is achieved by means of the Standard $k-\epsilon$ model with wall functions for the computation of the boundary layer in the near-wall regions, as suggested by Basha and Gopal [17]. They claim that most computational works employ two-equation turbulence models; the Standard $k-\epsilon$ being the best applicable model for modeling turbulence phenomena in engine simulations [11,13,15]. The PISO algorithm is employed to solve the pressure-velocity coupling and the second order upwind scheme is used for spatial discretization of the whole set of RANS and turbulence equations, as well as the bounded second order for time differencing, to obtain a good resolution [13]. The convergence of residuals was imposed to 10^{-5} for all quantities, while three revolutions of the crankshaft were necessary to correctly initialize the flow field into the ducts and to achieve a periodic behavior between two subsequent cycles [14]. The temporal discretization was assessed through a timestep sensitivity, by analyzing three different values of the angular timestep: 0.5° , 0.25° and 0.1° . Independent results were found with the finest resolution, therefore a final value of 0.25° was chosen as the angular increment per step.

The simulation domain is shown in Figure 1. Due to the symmetric geometry of the cylinder, only half of the entire model was considered. The symmetry plane is a cross section of the cylinder, the exhaust duct and the fifth transfer duct. As a result, five fluid sub-domains are included in the simulation. The inlet section is

placed at the crank-side end of the transfer ducts while the outlet section is placed at the end of the 90-degree bend exhaust duct. The communication between the sub-domains through the scavenge and exhaust ports and the cylinder is guaranteed by conservative sliding interfaces that connect common faces of adjacent zones.

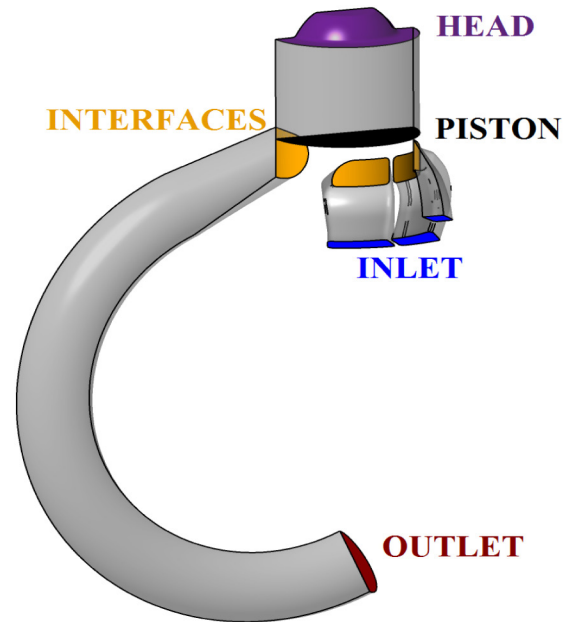


Figure 1. Simulation domain and boundary conditions.

Pressure and temperature inlet/outlet boundary conditions are imposed according to the results derived from specific 1-D simulations [13, 14, 15]. Since the simulations for the LPDI solution were carried out at a preliminary stage of the engine development, before the manufacturing of the first prototype, the numerical 1-D model was validated against experimental data collected for the indirect injection configuration. The PFI configuration was fine-tuned at the test bench in the first step of the project: it was obtained through the conversion of the carbureted engine to reach its best performance. Figure 2 shows the resulting characteristic curves.

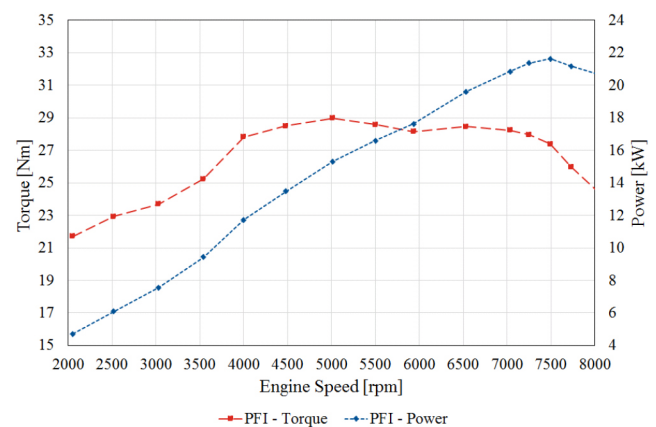


Figure 2. Torque and power curves for the PFI engine.

The solver is used to perform the analysis of the discharge, scavenge, fuel-air mixing and compression processes. The actual combustion process is not simulated but is modeled. The pressure and

temperature initial conditions for all domains are set before the exhaust port opens (EPO). Residual gas composition is imposed assuming a complete and stoichiometric combustion.

SPRAY MODELING

Spray droplets are calculated using the Discrete Phase Model (DPM). The droplets having the same velocity, diameter and temperature are treated as one parcel, and the motion is solved with a Lagrangian approach by tracking each parcel through the calculated flow field. A droplet in the spray forming process undergoes many physical phenomena during its path, including primary and secondary breakup, collision, etc.

In this study, spray atomization of the liquid core was not considered [13,14], since experimental data of droplet diameter distribution after the primary breakup were available. Figure 3 shows the measurements at 10 mm from the injection point in terms of Sauter Mean Diameter (SMD) and jet velocity (U), both reported as a function of the cone angle (δ) in a dimensionless form with respect to the averaged value.

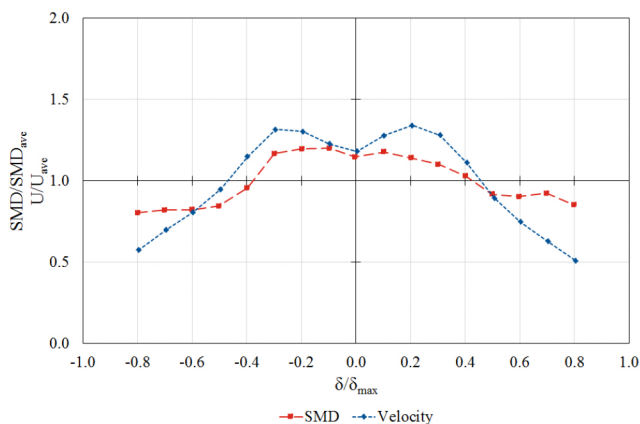


Figure 3. Experimental measurements at 10 mm from the injector tip: Sauter Mean Diameter and axial velocity.

The jet exit conditions were imposed using the Rosin-Rammler distribution [16], characterized by the most probable droplet size and a spread parameter. The suitability of a direct imposition of the diameter distribution is confirmed by the studies of Faeth et al. [18]. They observed that the ambient gas density has no effect on the primary breakup regime map in the range $104 < \rho_L / \rho_g < 7240$, where ρ_L and ρ_g are the liquid and gas densities, respectively. Since the injection process occurs mainly during the early compression and scavenge phases, the fresh charge density is relatively low and the density-ratio is in the range of 600-700.

The choice of the secondary droplet breakup model is based on the droplet Weber number [18], which is a measure of the ratio of the kinetic energy on impact to surface tension forces:

$$We = \frac{\rho_g U^2 d_d}{\sigma} \quad (1)$$

where U is the relative velocity between the liquid and the gas phases, d_d is the droplet diameter and σ is the droplet surface tension.

In the case of low pressure injection, i.e. 5 bar, the spray velocity is relatively low, determining values of We in a range bounded by 5-10. The Taylor Analogy Breakup (TAB) model was selected as the most recommended for low-Weber-number injections ($We < 100$), being well assessed for low-speed sprays [11,16]. It is based on Taylor's analogy between an oscillating and distorting droplet and a spring mass system.

COMPUTATIONAL GRID

The complexity of the computational domain, composed of five zones with different topologies (see Figure 1), imposes the definition of a separate mesh for each zone, namely the cylinder, the exhaust duct and the three transfer ducts (Figure 4).

A hexahedral structured grid was generated for the discretization of both the exhaust and the cylinder sub-domains. This solution guarantees better accuracy and stability, with cells that are oriented in the flow direction. Moreover, for the cylinder sub-domain, hexahedral elements are necessary to perform the mesh motion. In fact, the dynamic layering was implemented to add or to remove layers of cells adjacent to the moving piston boundary during the expansion and compression strokes, respectively. Conversely, all of the three transfer ducts have been meshed following a similar topology using an unstructured grid composed of tetrahedral elements; otherwise hexahedral elements would have led to an unacceptable skewed mesh.

A parametric analysis aiming at assessing the sensitivity of the computed solution to the level of mesh refinement and its outcomes is reported in the following. The baseline grid (*Mesh B*) has 5.0×10^5 cells, obtained imposing a mean cell size of 1.0 mm. This value has been hypothesized according to the discretization levels adopted by some of the authors in similar studies [19,20] and by other examples in literature [13,14]. To evaluate the solution sensitivity to mesh refinement, a coarser grid (*Mesh C*) and a finer grid (*Mesh F*) were obtained by varying the mean cells' size. The *Mesh C* features 2.0×10^5 elements while the *Mesh F* features 7.0×10^5 elements, corresponding to mean cell sizes of 0.75 mm and 1.5 mm, respectively. The curves of the relative pressure in the compression chamber during the scavenging process computed with the three different meshes are reported in Figure 5. The angular position of 180°CA corresponds to the bottom dead center (BDC). The discretization error has been minimized since negligible differences can be observed between *Mesh B* and *Mesh F* solutions. The grid-independency of the results was verified, confirming the selection of the *Mesh B* for the following analyses.

The accuracy of the numerical model in representing the physical problem of interest was validated at a later stage against experimental data collected at the test bench for the LPDI engine [9]. High frequency piezo-resistive and piezo-electric sensors were used to measure the instantaneous pressure inside the cylinder, the crankcase and exhaust ducts, which were compared with the CFD results. In