Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 81 (2015) 723 – 731

Energy Procedia

69th Conference of the Italian Thermal Engineering Association, ATI 2014

CFD Analysis of 2-Stroke Engines

Tommaso Savioli [a*]

[a] *University of Modena and Reggio Emilia, Strada Vignolese, 905 41125 Modena Italy*

Abstract

1d and 3d CFD simulation is a standard engineering practice on 4-stroke engines. However, the application to the 2-Stroke cycle is not so trivial, in particular the transition from detailed CFD-3d analyses to simplified 1d engine simulations. A first critical issue is the correlation between the instantaneous cylinder gas composition and the composition of the exhaust flow: such a correlation, required by the 1d software, is very difficult to be determined by experiments, so that CFD-3D analyses remain the only reliable source of information. These CFD analyses must be carefully designed, in order to be representative of the actual operating conditions of the engine, and to provide reliable results.
Another problem is the characterization of the ports discharge properties. Conventional experiments at a steady flow bench are affected by several approximations: first of all, flow patterns within the cylinder are quite different from the actual ones; moreover, the real flow is fully transient.
Finally, a 3-d analysis on a single whole cycle is not sufficient to accurately predict the evolution of the thermodynamic quantities, as the conditions of the charge at exhaust port opening changes from cycle to cycle, affecting all the internal processes. A robust and cost-effective multi-cycle simulation methodology is therefore required.
The paper describes a methodology to perform reliable CFD analyses on 2-Stroke engines, with the support of a conventional steady flow bench. This methodology is applied to a 2-Stroke engine prototype, for which a comprehensive set of experimental data is available.
The good agreement between simulation and experiments demonstrates the soundness of the proposed approach.

Peer-review under responsibility of the Scientific Committee of ATI 2014

Keywords: 2-Stroke-Engines, CFD simulation, scavenging, ports

1. Introduction

1d and 3d CFD simulation is a standard engineering practice on 4-stroke engines. However, the application to the 2-Stroke cycle is not so trivial, in particular the transition from CFD-3d analyses to 1d engine simulation. Such a

Corresponding author: Tommaso Savioli
Tel.: +39 059 2056152 *E-mail address:* Tommaso.savioli@unimore.it

Peer-review under responsibility of the Scientific Committee of ATI 2014
doi:10.1016/j.egypro.2015.12.078

transition is mandatory in order to predict the influence of design parameters on performance, over a wide range of operating conditions, in a limited elapse of time [[1],[2]].
A first critical issue is the modeling of the exhaust flow composition, which is the result of the mixing within the cylinder between fresh and spent charge. A correlation between the instantaneous cylinder gas composition and the one of the exhaust flow must be entered in the 1d software [[3]]: such a correlation is very difficult to be determined by experiments, so that CFD-3D analyses remain the only reliable source of information. These CFD analyses must be carefully designed, in order to be representative of the actual operating conditions of the engine, and to provide reliable results [[4]-[8]].
Another problem is the characterization of the discharge properties of the ports. Conventional experiments at a steady flow bench provide information that should be considered with care: first of all, flow patterns within the cylinder are quite different from the actual ones (as just an example, exhaust ports are generally tested blowing air through the cylinder head, while transfer ports are kept closed); moreover, the actual flow is fully transient.
Finally, a 3-d analysis on a single whole cycle is not sufficient to accurately predict the evolution of the thermodynamic quantities (in fact, the conditions of the charge at exhaust port opening changes from a cycle to the following one, affecting all the internal processes). As a consequence, a multi-cycle analysis is highly recommendable [[9],[10]].
The aim of the paper consists in defining an accurate and efficient methodology for the numerical simulation of the 2-Stroke engine cycle. This methodology is independent on the numerical tools, which are KIVA-4 (3-d analyses) and GT-Power (1d). The support of a standard steady flow bench is used to calibrate the 3d models, while the measures at a dynamometer bed provide the final evidence of the 1d model accuracy.
The proposed methodology is applied to a 2-Stroke engine prototype, for which a comprehensive set of experimental data is available [[11],[12],[13]]. This engine features standard transfer and exhaust ports, along with a set of auxiliary transfer ports controlled by a patented rotary valve [[14]], see figure 1. The valve gives the designer a further degree of freedom for defining the time-area diagram of the induction system.

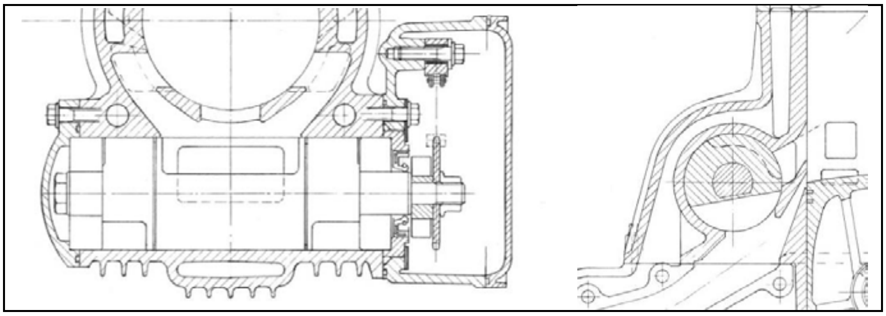


Figure 1: View of the rotary induction valve

The first stage of the study is the steady cfd-3d characterization of the ports discharge coefficients. The comparison with the equivalent experimental tests provides fundamental information for the calibration of the 3d models, in particular the refinement of the grid. Furthermore, the port discharge coefficients are used as input data in the GT-Power model [[3]].
The second stage of the study is the scavenging analysis, performed by means of KIVA-4 [0]. The goals of this numerical activity are: 1) find a correlation between the instantaneous composition of the charge within the cylinder and the composition of the exhaust flow; 2) determinate the scavenging parameters such as scavenging efficiency and trapping efficiency as a function of delivery ratio; 3) calculate the turbulence parameters within the cylinder
These results are fundamental for setting the Gt-Power model, in order to get a reliable simulation of scavenging and combustion (in the predictive combustion model, the turbulent combustion speed calculation is based on the assessment of turbulence intensity).
The analysis is performed at different engine speeds (2000, 3000 and 4500 rpm), with different pressure ratio across the cylinder (1.1, 1.2, 1.3, 1.4 and 1.5)
The third stage is the 3-d simulation of the whole engine cycle for multiple cycles. In this case, boundary conditions are not constant, but they are provided by previous GT-Power analyses. The results of the multi-cycle 3-d

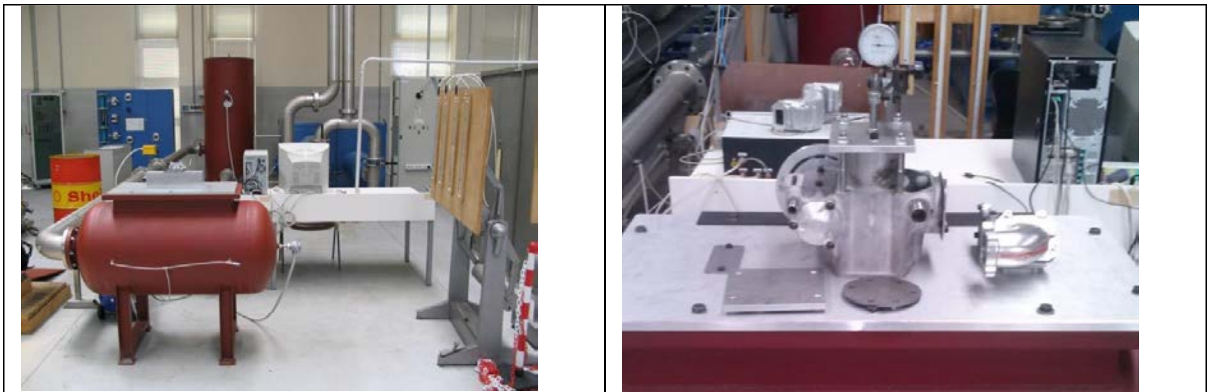
simulations are used for a finer calibration of the GT-Power model.
Finally, the last step is the 1-d engine simulation at full load and the comparison with the experimental results obtained on the prototype at the dynamometer bench.

2. Steady Flow analyses

2.1 Experimental setup

The experimental tests presented in this paper are performed at the steady flow bench installed at the Engineering Department “Enzo Ferrari”, in Modena. The system, visible in Figure 2 a), is fully designed and built in-house. A flow rate up to 1500 Nm^3/hr can be generated by a 5-stage blower, powered by an electric motor of 55 kW. The flow is regulated by governing through an inverter the speed of the motor. At the blower outlet, a system of 4 valves can swap the flow direction, and a 1.5 m^3 tank is adopted to damp the flow oscillations. Another smaller tank (set-up tank) is used to achieve still air conditions below the mock-up. The flow rate is measured by a Pitot probe, installed in a straight pipe between the damping and the set-up tanks; the Pitot differential pressure is measured by a piezo-resistive pressure gauge.
The cylinder mock-up is installed on a holed plate, bolted on the set-up tank (Figure 2 b)). The piston can be manually shifted by screwing a threaded rod on a fixed plate. The lift is measured by a micrometer gauge, having a resolution of 1/50 mm. A second piezo-resistive pressure gauge and an electric thermometer measure the differential pressure and the temperature in the set-up tank. Finally, a thermometer and a barometer are used to get the environmental conditions.
All the sensors provide DC current signals, digitalized by a data acquisition board, linked to a PC. The data are processed, visualized and saved on the PC using a customized VBA (Visual Basic for Application) program linked to MS Excel.
Three different test batches are performed:

1) Conventional transfer ports: 5 openings (2,3,4,5 and 6 mm), 4 pressure gradients (5, 10, 15 and 20 kPa);
2) Conventional and auxiliary transfer ports: 6 piston and rotary valve positions, corresponding to 6 conventional transfer port openings (0, 2,3,4,5 and 6 mm), 4 pressure gradients (5, 10, 15 and 20 kPa);
3) Exhaust ports: 8 openings (2.0, 3.9, 8.0, 11.8, 15.9, 19.8, 22.4 and 24.4 mm), 4 pressure gradients (2.5, 5, 7.5 and 10 kPa).



a)	b)

Figure 2: the steady flow bench facility.

2.2 CFD simulations

The steady cfd-3d characterization of the ports are performed by using the KIVA 4 code. The computational grid has been built devoting particular care to accurately model the ports height, in order to maintain the same timing of the real geometry. The computational grid is shown in Figure 3 a), where the set of auxiliary transfer ports above the conventional ones is clearly visible on the right. The variation of the orifice area controlled by the rotary induction valve is modeled by means of one layer of cells at the inlet boundary, that is progressively switched from wall to fluid [[11]].

2 different configurations are simulated in order to characterize both transfer (conventional and auxiliary) and exhaust ports. In the first case, see Figure 3 b), the cylinder head surface is set as an outlet boundary, while the exhaust duct is kept closed; in the second case, shown in Figure 4 c), the cylinder head surface is set as an inlet boundary and the transfer ducts are kept closed.

For the transfer ports characterization, at first the auxiliary ports are kept closed, while the conventional ports opening is varied between a minimum and a maximum value; then the tests are carried out with both sets of ports. The flow rate through the auxiliary ports is assumed to be the difference between the flow rates measured in the two cases (superposition of effects). The tests are carried out shifting the piston between 35 cad BBDC and 80 cad ABDC, at a 5 cad step, applying the same pressure gradients adopted in the experiments.

2.3 Results

The comparison between steady flow experiments and CFD-3D results, shown in figures 4-6, demonstrates the accuracy and the robustness of the numerical approach. From both simulation and experiments it was found that the discharge coefficients are quite independent on the pressure gradient, so that a single average value can be plotted for each piston position.

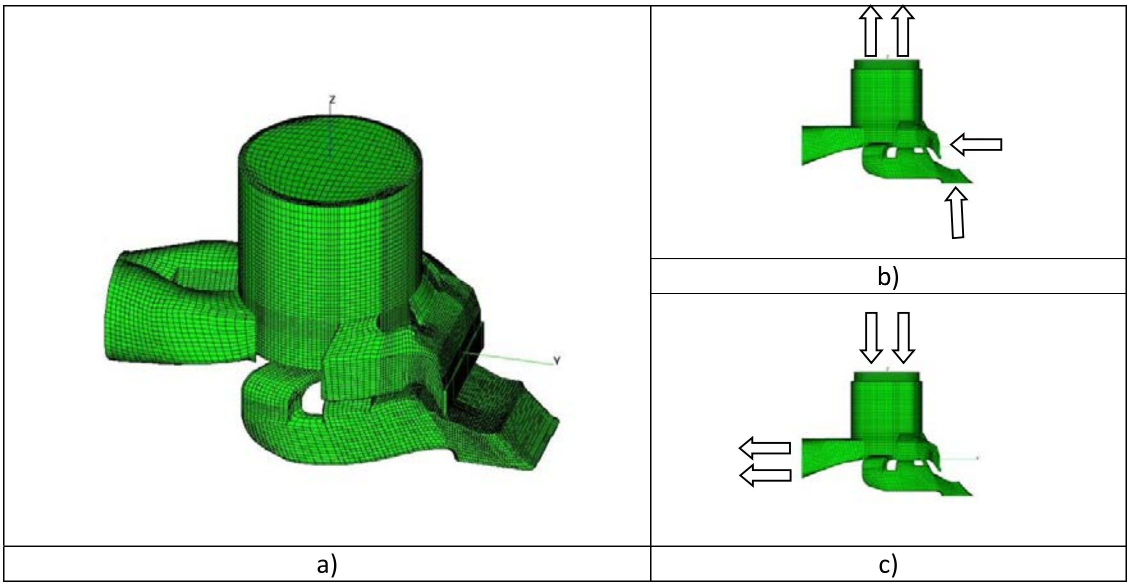
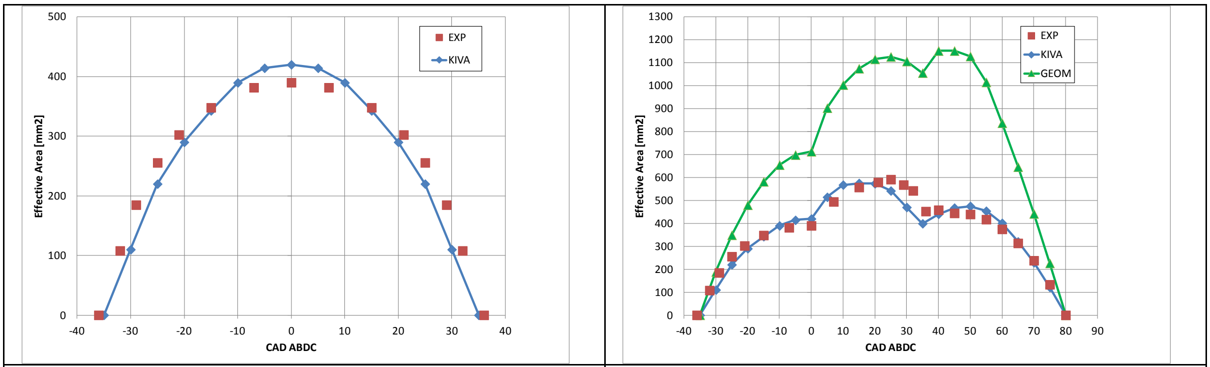
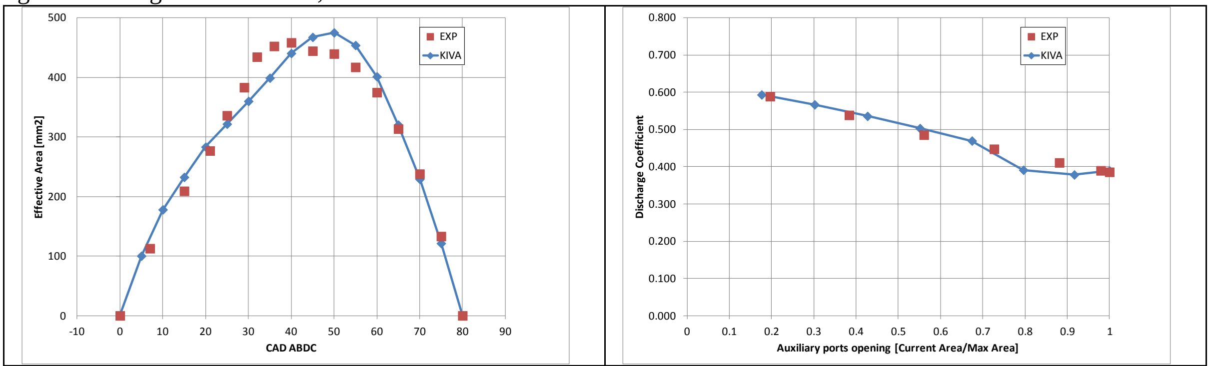


Figure 3: the computational grid



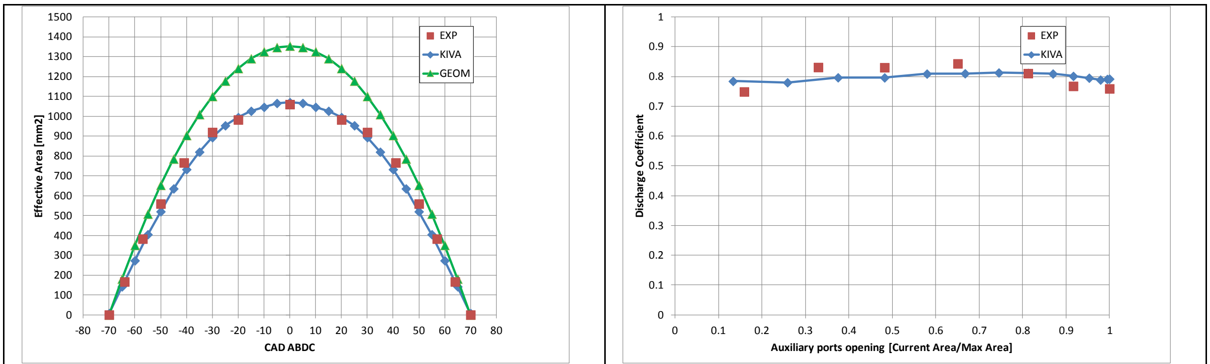
a) b)

Figure 4: a) Effective area of the conventional transfer ports plotted as a function of crank angle. Red points: experimental values; blue curve: CFD-3d simulation; b) Total effective area of transfer ports (conventional and auxiliary) plotted as a function of crank angle. Red points: experimental values; blue curve: CFD-3d simulation; green curve: geometrical area;



a) b)

Figure 5: a) Effective area of the auxiliary transfer ports plotted as a function of crank angle. Red points: experimental values; blue curve: CFD-3d simulation; b) Discharge coefficients of the auxiliary transfer ports plotted as a function of opening (current valve geometric area to max. geometric area). Red curve: experimental values; blue curve: CFD-3d simulation



a) b)

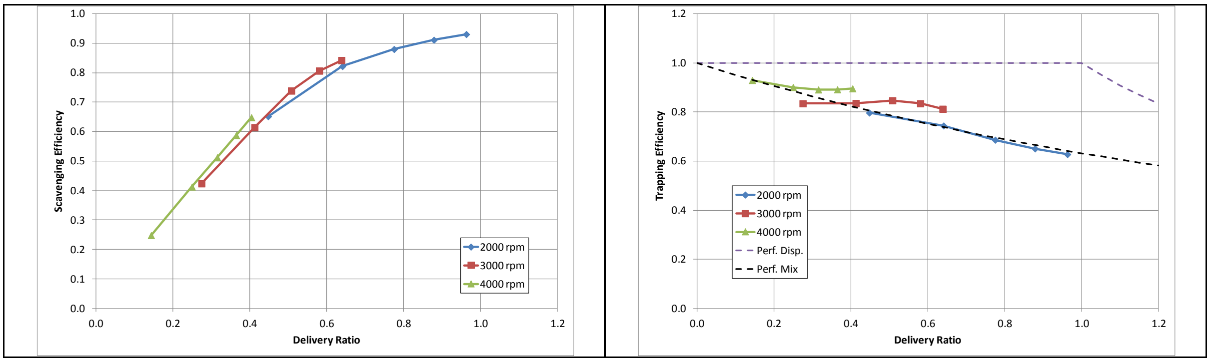
Figure 6: a) Effective area of the exhaust port plotted as a function of crank angle. Red points: experiments, blue curve: simulation; green curve: geometric area. b) Discharge coefficients of the exhaust ports plotted as a function of port opening (current port lift to max. port lift). Red curve: experimental values; blue curve: CFD-3d simulation

3. Scavenging analyses

In the scavenging analysis, the computational domain is made up of cylinder, head, piston and ports; calculations start just before exhaust port opening and stop at 80° ABDC; initial conditions are provided by GT-Power calculations, while pressures at the boundaries are held constant. The analysis is performed at different engine speeds (2000, 3000 and 4500 rpm), with different pressure ratio across the cylinder (1.1, 1.2, 1.3, 1.4 and 1.5). Pure Oxygen and pure Nitrogen are used to represent fresh charge and residuals, respectively [[5]].

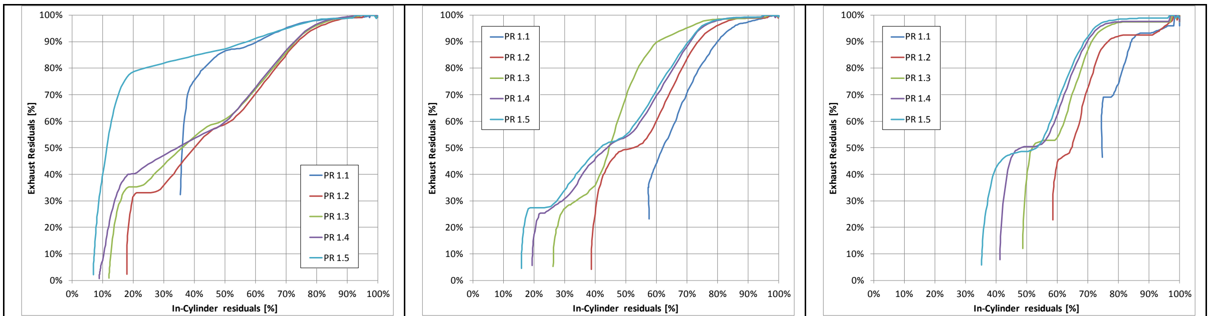
One of the most interesting output of the 3D-CFD calculations is the assessment of the scavenging process quality, i.e. the dependence of Trapping and Scavenging Efficiency on Delivery Ratio. The Trapping Efficiency (TE) is defined as the ratio of the mass of fresh air retained to the mass of fresh air delivered; the Scavenging Efficiency (SE) is the ratio of the mass of fresh charge retained to the mass of cylinder charge; the Delivery Ratio (DR) is the ratio of the mass of fresh charge delivered to the reference mass; finally, the reference mass is calculated considering the average delivery density and the total displaced volume.

Figure 7 shows the parameters defined above for 3 engine speeds and 5 different pressure ratios across the cylinder. As expected, as engine speed decreases, for a given PR, DR increases (longer scavenging time). Keeping constant engine speed, DR is about proportional to PR. Combining all the 15 cases, it is possible to analyze the scavenging process over a wide range of DR, from 0.15 to 0.95. Figure 7 a) shows that SE features are quite independent on speed even if a slight improvement may be observed at high speed. TE reported in Figure 7 b) confirms this trend and shows that at high DR the scavenging process is quite close to a perfect mixing.



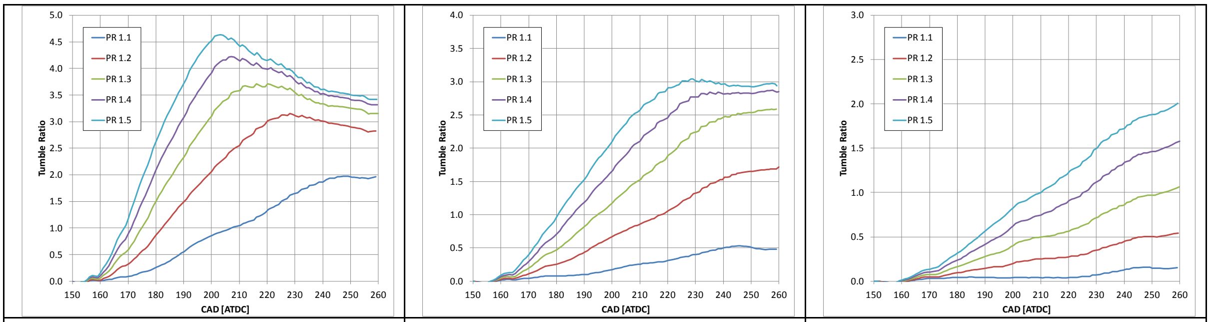
a) b)

Figure 7: Scavenging Efficiency (a) and Trapping Efficiency (b) plotted as a function of Delivery Ratio for different engine rotational speeds and different Pressure Ratio



a) b) c)

Figure 8: Exhaust residuals plotted as a function of in-cylinder residual fraction for different Pressure Ratio (1.1, 1.2, 1.3, 1.4 and 1.5) at a) 2000 rpm, b) 3000 rpm and c) 4500 rpm



a)	b)	c)

Figure 9: Tumble Ratio plotted as a function of crank angle for different Pressure Ratio (1.1, 1.2, 1.3, 1.4 and 1.5) at a) 2000 rpm, b) 3000 rpm and c) 4500 rpm

Figure 8 reports the mass fraction of fresh charge in the exhaust flow leaving the cylinder as a function of the in-cylinder residual fraction. These results must be entered as input data in the GT-Power simulations, and they are also useful to assess the scavenging quality. A good scavenging requires that all the curves stay over the perfect mixing line (the closer to perfect displacement, the better)

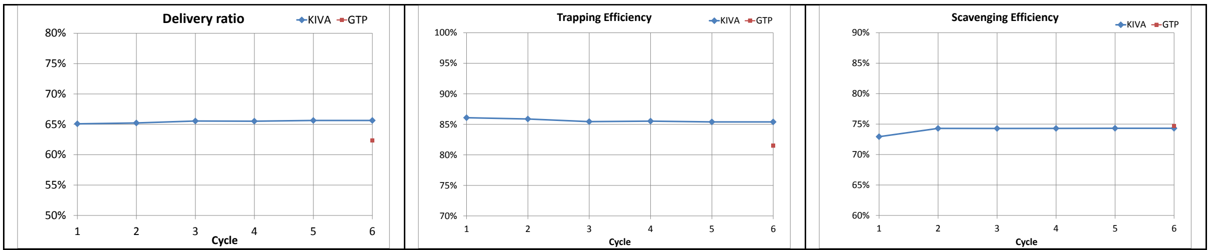
The values of Tumble Ratio, visible in Figure 9, are necessary to set the GT-Power simplified turbulence model, in order to predict combustion rates. The macro-turbulence intensity is about proportional to pressure ratio, and it decreases as engine speed increases.

4. Multi-cycle analyses

The number of CFD-3D cycles required to reach convergence strongly depends on the accuracy of the GT-Power engine model, proving the initial and boundary conditions. Thanks to the previous characterizations, it was found that the 3D results are reasonably stable from just the third cycle.

Figure 10 presents the trends of DR, TE and SE at 4500 rpm, full load. It may be noticed that the final values are very close to the ones predicted by GT-Power.

A further confirmation of the 1d model accuracy comes from the comparison with KIVA in terms of in-cylinder pressure, figure 11.



a)	b)	c)

Figure 10: Delivery ratio, trapping efficiency and scavenging efficiency plotted as a function of the cycle number at 4500 rpm, full load. Gt-Power values are also plotted, as a reference.

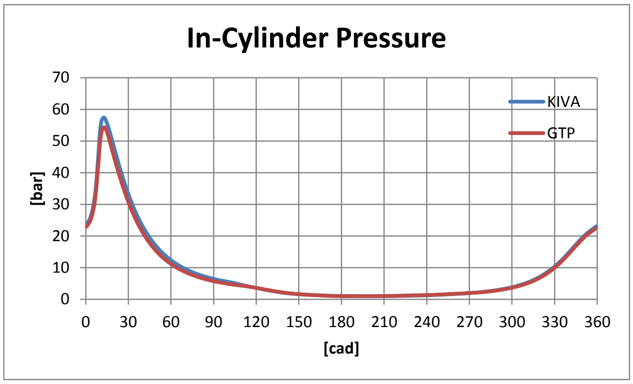


Figure 11: In-cylinder pressure predicted by the multi-cycle 3d analysis and by the GT-Power model

5. Validation at the dynamometer bench

The final step of the study consists in the engine characterization at the dynamometer bench. The experimental measures have been taken on a prototype installed at the BRC laboratories in Cherasco, Italy [[13]]. The comparison with the corresponding GT-Power results is the definitive evidence of the proper set-up of the 1d model, and of the effectiveness of the integrated experimental and numerical approach.

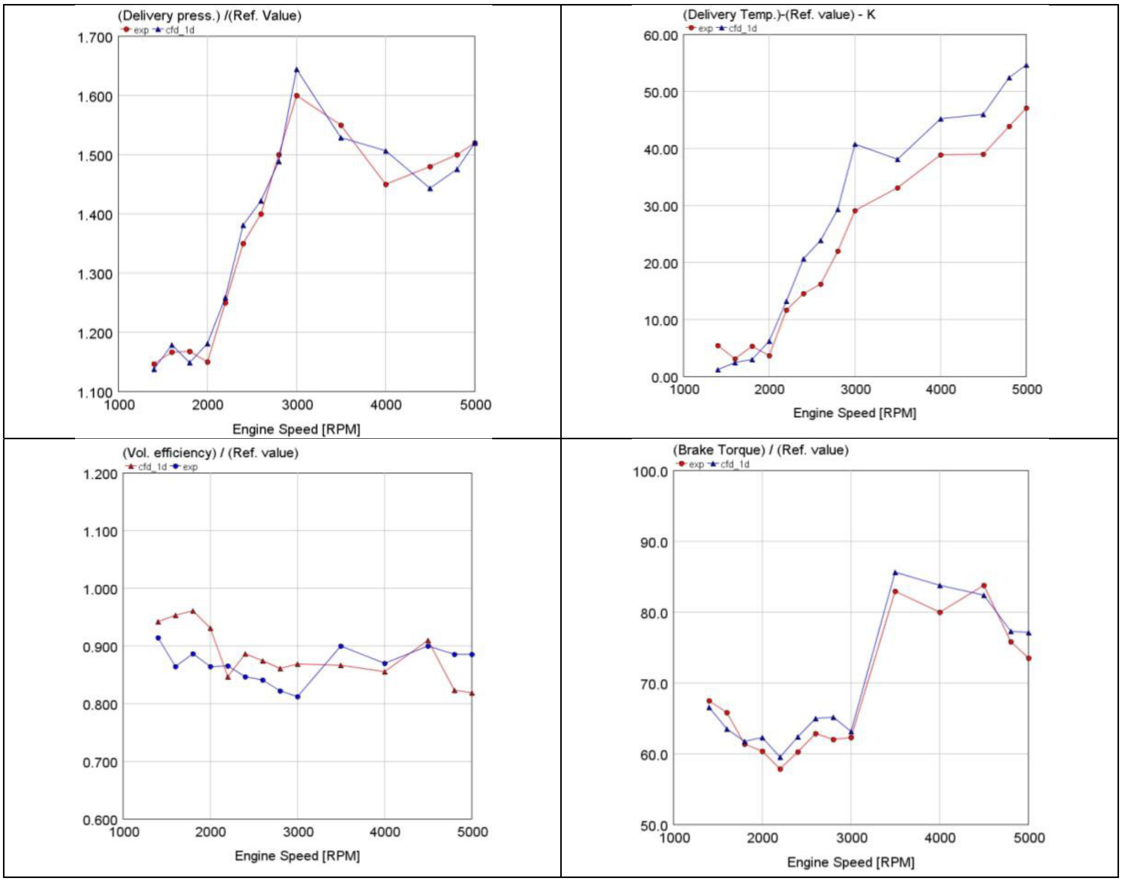


Figure 12: Comparison between GT-Power results and measurements on the 2-S prototype by BRC, at full load.

6. Conclusion

The paper reviews an integrated experimental and numerical approach for the analysis of 2-Stroke engines. The methodology is applied to a GDI single cylinder prototype, featuring a patented rotary valve.

The experimental facilities required for developing the proposed method are a standard steady flow bench and a dynamometer bed. The former is used to tune the CFD-3d models, the latter is employed for the final assessment of the accuracy of the cfd-1d tool. In order to achieve a good quality of engine simulation, it is necessary to import information from the 3d analyses, in particular about port discharge properties and scavenging features. As for 4-Stroke engines, also the details about combustion and heat transfer give a fundamental contribution.

References

[1] Heywood, J.B. and Sher, E. "The Two Stroke Cycle Engine – Its Development, Operation, and Design", Taylor & Francis, Inc ISBN 1-56032-831-2

[2] Blair, G. and Kenny, R., "Further Developments in Scavenging Analysis for Two-Cycle Engines," SAE Technical Paper 800038, 1980, doi: 10.4271/800038.

[3] Gamma Technologies, "GT-Power v7.4 user manual", 2013

[4] Mattarelli, E., "Virtual Design of a Novel 2-Stroke HSDI Diesel Engine", International Journal of Engine Research, PE Publishing, Issue June 2009, Vol. 10 No 3 ISSN 1468-0874, pp. 175-193

[5] Rinaldini, C.A., Mattarelli, E., and Golovitchev, V., "CFD Analyses on 2-Stroke High Speed Diesel Engines," SAE Int. J. Engines 4(2):2240-2256, 2011, doi:10.4271/2011-24-0016.

[6] De Marco, C., Mattarelli, E., Paltrinieri, F., and Rinaldini, C., "A New Combustion System for 2-Stroke HSDI Diesel Engines," SAE Technical Paper 2007-01-1255, 2007, doi:10.4271/2007-01-1255

[7] Mattarelli, E., Rinaldini, C.A., and Wilksch, M., "2-Stroke High Speed Diesel Engines for Light Aircraft," SAE Int. J. Engines 4(2):2338-2360, 2011, doi:10.4271/2011-24-0089.

[8] Mattarelli, E., Paltrinieri, F., Perini, F., Rinaldini, C.A. and Wilksch, M., "2-Stroke Diesel Engine for Light Aircraft: IDI vs. DI Combustion Systems," SAE Technical Paper 2010-01-2147, 2010, doi:10.4271/2010-01-2147

[9] E. Mattarelli, S. Fontanesi, V. Gagliardi, S. Malaguti. "Multidimensional Cycle Analysis on a Novel 2-Stroke HSDI Diesel Engine", SAE Paper 2007-01-0161. 2007

[10] G. Cantore, S. Fontanesi, S. Malaguti, E. Mattarelli, "CFD-3D Multi-Cycle Analysis on a New 2-Stroke HSDI Diesel Engine", SAE paper 2009-01-0707. 2009

[11] Mattarelli, E., Rinaldini, C., Cantore, G., and Baldini, P., "2-Stroke Externally Scavenged Engines for Range Extender Applications," SAE Technical Paper 2012-01-1022, 2012, doi:10.4271/2012-01-1022.

[12] Mattarelli, E., Rinaldini, C., and Cantore, G., "Comparison between a Diesel and a New 2-Stroke GDI Engine on a Series Hybrid Passenger Car," SAE Technical Paper 2013-24-0085, 2013, doi:10.4271/2013-24-0085

[13] Mattarelli, E., Rinaldini, C., and Baldini, P., "Modeling and Experimental Investigation of a 2-Stroke GDI Engine for Range Extender Applications," SAE Technical Paper 2014-01-1672, 2014, doi:10.4271/2014-01-1672

[14] Baldini, P., "2-Stroke Engine with Low Consumption and Low Emission", International Patent PCT/IT2010/000057, Publication N° WO/2011/101878, on August 25, 2011.

Torres, D. J. "KIVA-4 manual." Los Alamos National laboratory Theoretical division (2006).