Three Dimensional Modeling of Flow and Mixture Preparation in a Two Stroke Engine

A.Fabre and C.Ferreira

P.S.A. Etudes et Recherches
Unite de Recherche Moteurs et Propulsion
Centre Technique Citroen
Chemin Vicinal N°2
78140 Velizy Villacoublay
France

ABSTRACT

This document describes the action we have taken to understand more fully the nature and the origin of the phenomena observed during the development of the two-stroke IAPAC engine. The characterisation of the internal aerodynamics and of the mixing of the gases is accomplished with the help of a specially adapted version of the KIVA code. The instantaneous flows through various orifices are defined following the treatment of a series of pressure measurements taken on an engine test bench at different points of the configuration. This treatment calls on a 0D simulation of the whole of the geometry. The analyses effected allow us to judge the quality of the scavenging loop and to approach the final stratification of the mixture. It was also possible to compare the operation at various speeds and loads; additionally, a change in the geometry of our engine has been studied.

INTRODUCTION

The two-stroke engine has over the past few years become the object of renewed interest for automobile application. Possessing a high specific power, it proves seductive due to its lightness and compact dimensions. Additionally, its fuel consumption at light load should be lower than the one of a four stroke engine, due to its small level of pumping losses and frictional losses. But this engine still suffers from its tendency to generate a high quantity of unburnt hydrocarbons and its poor operational stability at low load.

Such disadvantages appear today surmountable, especially with a new generation of two-stroke engine studied in collaboration with Institut Français du Pétrole. By dissociating the scavenging of the burnt gas from the fuel admission function, i.e. the Compressed Air-Assited Fuel Injection (IAPAC) technology proposed by IFP, an important reduction of unburnt hydrocarbon emissions has been obtained. In order to achieve a satisfactory stability for all operating conditions, we now associate numerical simulations with engine test bench experiments in the manner which will be described in this article.

REMINDER OF IAPAC TECHNOLOGY

The principle of IAPAC and its advantages have already been described in various publications (1,2). The main points can be summarized as follows :

A rich mixture is prepared in a surge tank connected to the crankcase and filled with air during the expansion stroke. The pressure is then maintained by a reed valve, which isolates it from the crankcase when the pressure drops.

The transfer of the mixture to the cylinder, made possible by the pressure reached, is then controlled by a poppet valve; the latter is actuated rather late in the cycle (after BDC).

Thanks to this concept, the major part of the scavenging phase is effected in the absence of gasoline : the air coming from the inlet transfer ports is pure air which will henceforth be called fresh gas.

However, like all two-stroke engines, this suffers from a reduction in overall efficiency of scavenging at part load. It is advisable therefore to use the available air to best advantage, by exercising a favourable influence on the internal aerodynamics resulting from the geometry; such a study has begun on an engine whose main features are presented on table 1.

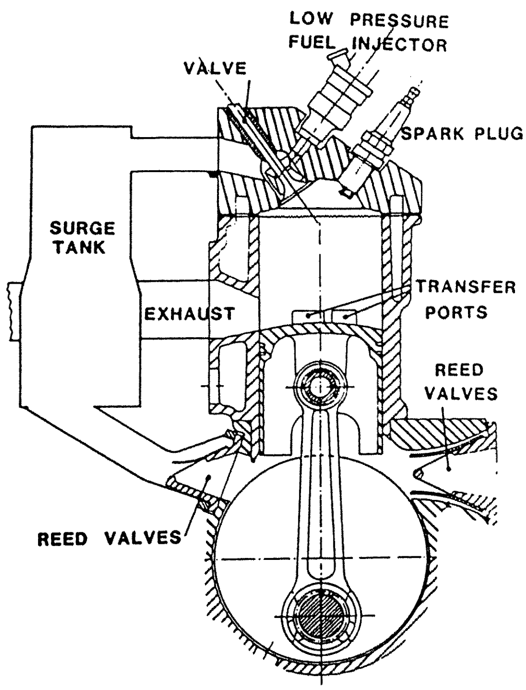

Fig. 1 The IAPAC Two-Stroke Engine

Table 1 Main features of the geometry

Bore x Stroke	0.084m x 0.084m		
Trapped compression ratio	8.5		
Crankcase / Surge tank volume	$10^{-3} m^3 / 10^{-3} m^3$		
Number of transfer ports	3		
Cylinder ports and valve timing (CA) :			
Exhaust opens	76 BBDC	closes	76 ABDC
Transfer opens	54 BBDC	closes	54 ABDC
Valve opens	15 ABDC	closes	85 BTDC

ADVANTAGES OF THE MULTIDIMENSIONAL SIMULATION AND PRESENTATION OF THE CODE.

Multidimensional calculations have already been widely used for four-stroke spark ignition engines (3) or diesels (4) ; the correlations achieved with the ADL measurements have shown its capability to reconstitute in a satisfactory manner the nature of the flows (3). This technique being hence able to make a significant contribution to the advancement of our work, we resorted to the KIVA code (5,6). The version which we use contains some modifications with respect to that initially proposed by the Los Alamos National Laboratory. We will describe them here, but limiting ourselves to the aerodynamic models which are the only ones concerned by this study.

Turbulence Model and Law of the Wall

The turbulence model delivered with the KIVA code was of the SGS type. From our experience, we now prefer a K-ε type formula; the compressibility effects are taken into account in a classic way(7). Besides, owing to its deficiency where the anisotropy of the flow is more pronounced and where the phenomena linked to the molecular viscosity assume a greater importance, this turbulence model is associated with a law of the wall (8).

Mesh and Yielding Ports

Taking into account its shape, our geometry required that we use a cartesian mesh (figure 2), which did not pose any particular problem, as the code was already interfaced with a commercial mesh generator. To simulate the fluid movement at various orifices, we generalised a treatment which gave us satisfaction when it was applied to the inlet valve of a four-stroke engine. The principle of this treatment is as follows :

Definition of yielding nodes. The portions of the boundary where a material exchange is likely to happen (valve lift, ports) are specified as data. The lines which define the contours of this orifice are, at each moment, identified and the mesh is obliged to follow theses lines. All the nodes eventually located inside the windows so defined are then declared yielding.

Assignment of speeds. According to what we know about the flow, we may either use speed profiles with constant modulus or take into account some variations with node's position and port's current area. In the same way, the orientation chosen for each node may only result from the design of the pipes or follow some more accurate data.

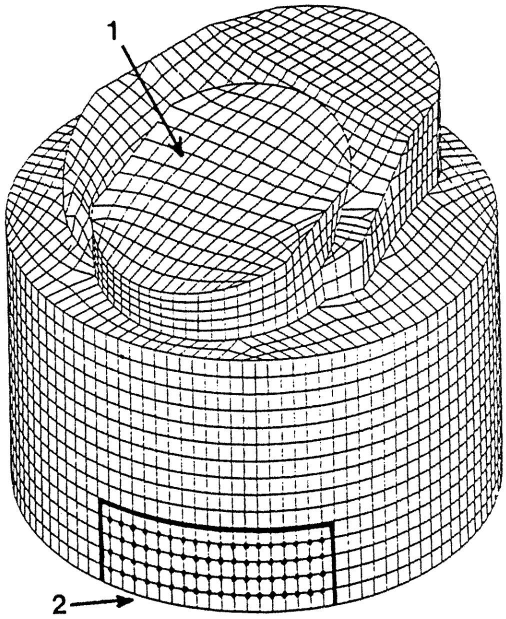

Fig. 2 The Mesh

(1) IAPAC valve

(2) Rear transfer port

(•) A yielding node

Assignment of other quantities. Each of the nodes so defined belongs to eight cells ; four of them correspond to cells which are really fluid (those situated inside the cylinder). The four others are also assimilated to fluid cells, but we impose on them values for k, ε, u and ρ at each step in time.

This algorithm does not raise any special difficulty, apart from those related to the management of the mesh, a point which would be too long to develop here.

THE 0D SIMULATION AND FLOW CALCULATION

Given these various arrangements, the 3D code proved to be perfectly operational for our problems. However, before starting any serious exploitation, it was necessary to derive suitable boundary conditions, for the various modes of operation. To obtain the flows and thermodynamic conditions prevailing at the various orifices, we could have had recourse to a 1D modelling of the whole of the geometry (including the inlet and exhaust lines), but some pressure measurements obtained on the engine test bench were available and we restricted ourselves to a simplified 0D simulation.

Presentation of the 0D Simulation

Role : This simulation permits the evaluation of mass exchanges between the different parts of the engine which are all taken into account, so obtaining the values of flow, speeds and temperatures at the openings of the cylinder ; these will serve as boundary conditions for the calculation with KIVA.

Principle. This is an iterative computation which is made on the basis of relatively arbitrary initial conditions and which only assumes as boundary conditions the pressure in the exhaust pipe (obtained experimentally) and the pressure in the inlet manifold which is assumed to be equal to atmospheric pressure. Each calculation step comprises two stages. During the first one, we consider two by two and successively the different parts of the engine likely to exchange material (for instance cylinder/crankcase via transfer ports); each time we evaluate the flow speed which would be established in the duct assuming stationnary conditions and we deduce a value of flow rate. The latter is then adjusted with a constant factor which accounts for the limited validity of our assumptions.

As a second step, a thermodynamic account is drawn up for each cavity considered in isolation. This account integrates not only the material exchanges evaluated before, but also the movement of the piston and its associated work, and finally the energy transfers linked to the combustion or thermal losses.

Fixing the parameters of the simulation. This stage is of course as important as the elaboration of the simulation itself. To work it out, we have equipped our engine test bench with a device which allowed us to take numeric pressure readings simultaneously on four channels, for each degree of crankshaft rotation during 170 cycles. As well as the exhaust port pressure values, we have been able to record those from three other pressure transducers located at the cylinder head, in the crankcase and in the surge tank. For each operating condition, those three measurements were used to fit the parameters of the simulation. The level of agreement is illustrated on figure 3.

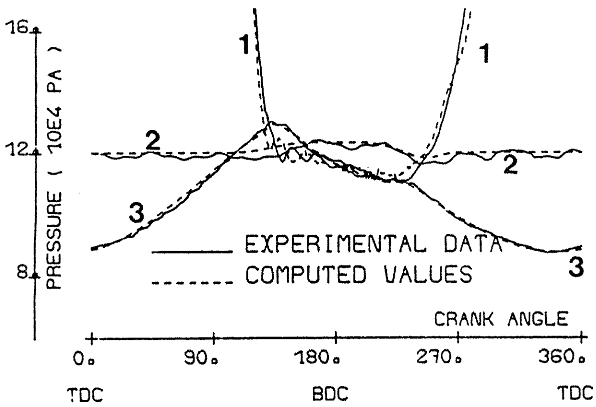

Fig. 3 Mean Experimental Pressures against Simulation Results

(1) In-cylinder pressure

(2) Surge tank pressure

(3) Crankcase pressure

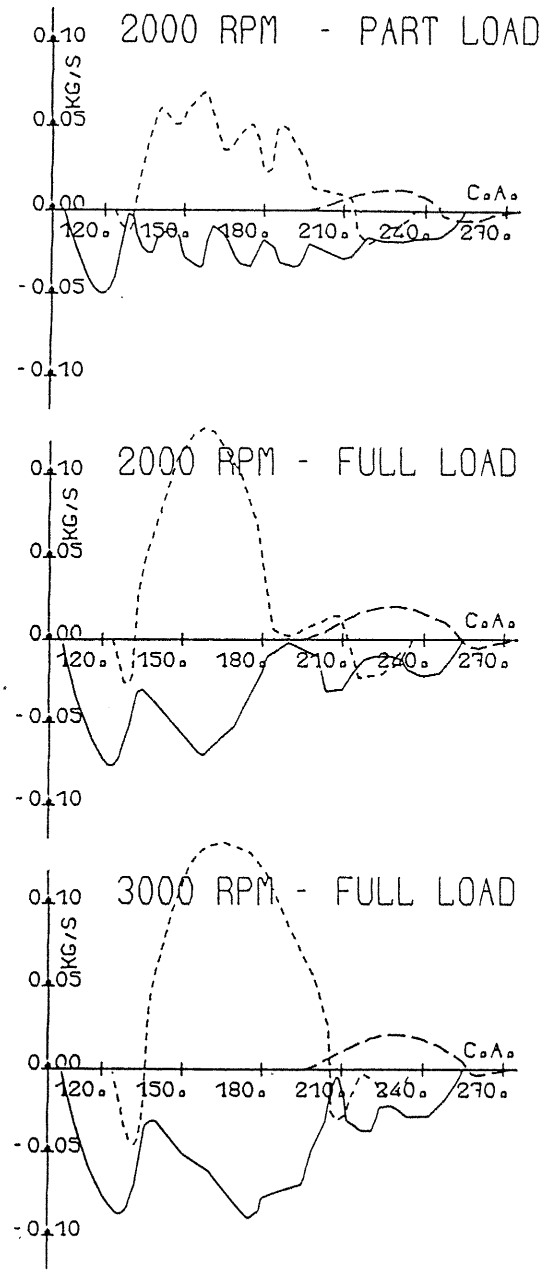

Fig. 4 Flow Rates as Obtained with 0D Simulation

- – – – through the IAPAC valve
- ------- through the intake ports
- ——— through the exhaust port

Results Connected with the Various Operating Conditions

Three operating points have been successively considered : one at 3000 RPM (full load), two at 2000 RPM - the first one at full load, the other at half load. Thanks to our simulation, we have obtained the flow curves shown on figure 4. The air consumptions which we have also been able to deduce from our results are in relatively close agreement with the values measured on the engine test bench (table 2).

Looking at these first results, it may be noted immediately that the behaviours at full load are fairly similar, although with an earlier scavenging completion in the 2000 RPM case; it may also be remarked that the maximum speed of the flow issuing from the inlet transfers is, as a first approximation, proportional to the load.

Table 2 Air consumption (units 10^{-3} kg/s)

	2000 RPM Full Load	2000 RPM Half Load	3000RPM Full Load
Experimental data	14.7	7.9	19.5
Computed values	14.	6.4	19.7

UTILISATION OF THE 3D CODE TO STUDY THE EFFECTS OF ENGINE SPEED AND LOAD

3D Calculation Conditions

The comparison has been further conducted on the basis of 3D simulation results. These were carried out in the following conditions : start of calculation at 240 BTDC (inlet ports still closed) with uniform temperatures and pressures inside the geometry. The components of initial velocity are null except the axial one : its value decreases linearly from the piston to the cylinder head. The initial turbulent kinetic energy is uniform and the dissipation rate of the turbulence is inversely proportional to the distance to the nearest wall. As far as velocity fields at the ports are concerned, this first evaluation has been made with rather basic hypotheses : we assumed that the velocity has a uniform modulus over the whole section and that its orientation for each node is defined by interpolation between the directions followed by the ducts in each corner of the orifice.

Results : Characterisation of the Scavenging Loop and Final State of the Mixture

Our purpose is to characterize the final state of the mixture and to understand what succession of phenomena brought such a result. The mass variations of the various gases enclosed in the cylinder are displayed on figure 5 for the three operating points. Besides, among all the analyses conducted, we have selected :

The contour-plots of axial velocity. Our first judgement is based on the distribution of the ascending and descending flows inside the cylinder (figure 6). Displayed at the same crank angle for several horizontal planes, these sections did not show up localised recirculation, and we considered this as a positive point. However, a slight tendency is seen at part load where the interface appears more deformed.

The contour-plots of residual gas concentrations. The investigation is carried out on a vertical section cutting the rear transfer and the exhaust ports (figure 7). It appeared that the path of fresh gases is essentially the same in the three cases (even at part load, the scavenging loop does not seem to be shortened), but with progression speeds which are quite different. Consequently, at part load, a high proportion of residual gases remains trapped in the cylinder (table 3).

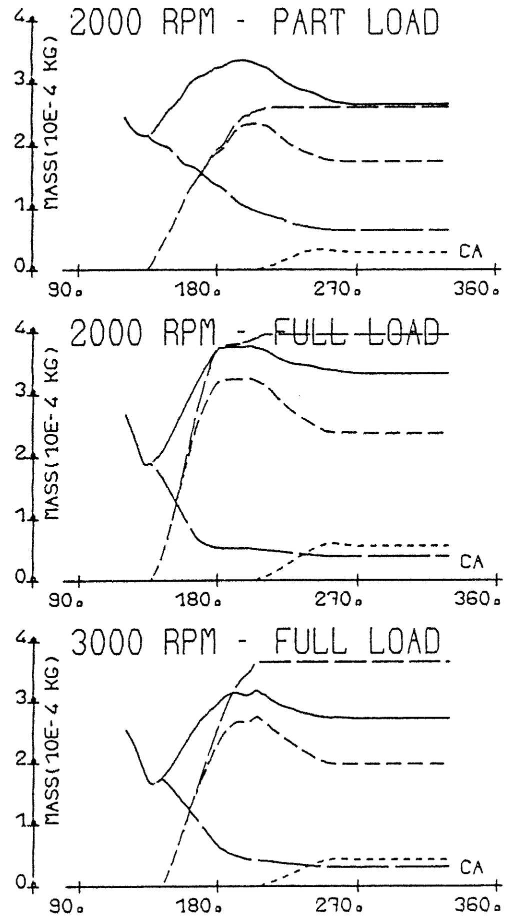

Fig. 5 Gases Flow Rates and the Masses Variation in the Cylinder

TOTAL IN-CYLINDER MASS
IN-CYLINDER RESIDUAL GASES
TOTAL AIR FROM CRANK-CASE
TRAPPED AIR FROM CRANK-CASE
TRAPPED AIR FROM SURGE TANK

Table 3 Final mixture composition as seen by the simulation

		Fresh Gas	Rich Gas	Residual Gas
2000 RPM	Part Load	65.3%	10.4%	24.3%
2000 RPM	Full Load	71.4%	16.8%	11.8%
3000 RPM	Full Load	72.5%	15.8%	11.7%

Taking into account the aerodynamic movement which persists in the chamber after the exhaust closes (figure 8), the simulation even predicts a higher than average proportion of residual gases (to the detriment of rich gases) in the spark plug zone at the moment of ignition.

The first elements drawn from this are interesting and give us indications in order to improve part load operation.

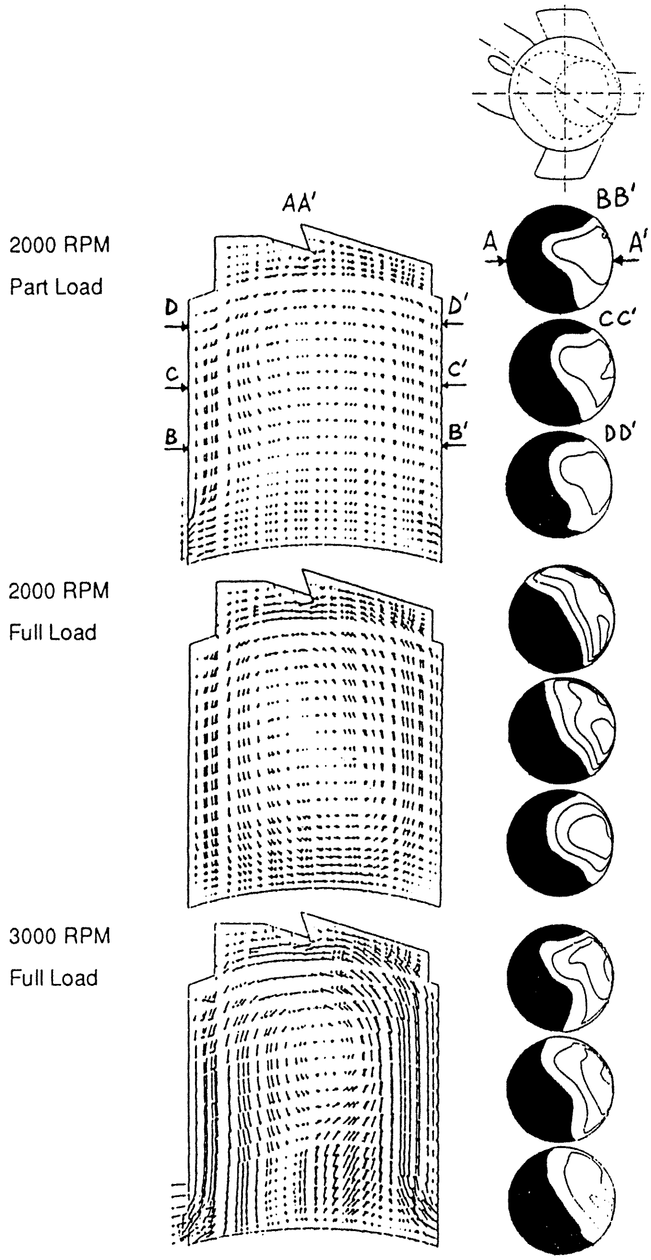

Fig. 6 Velocity Field Visualisation at 170 BTDC

Upward gas flow

Downward gas flow

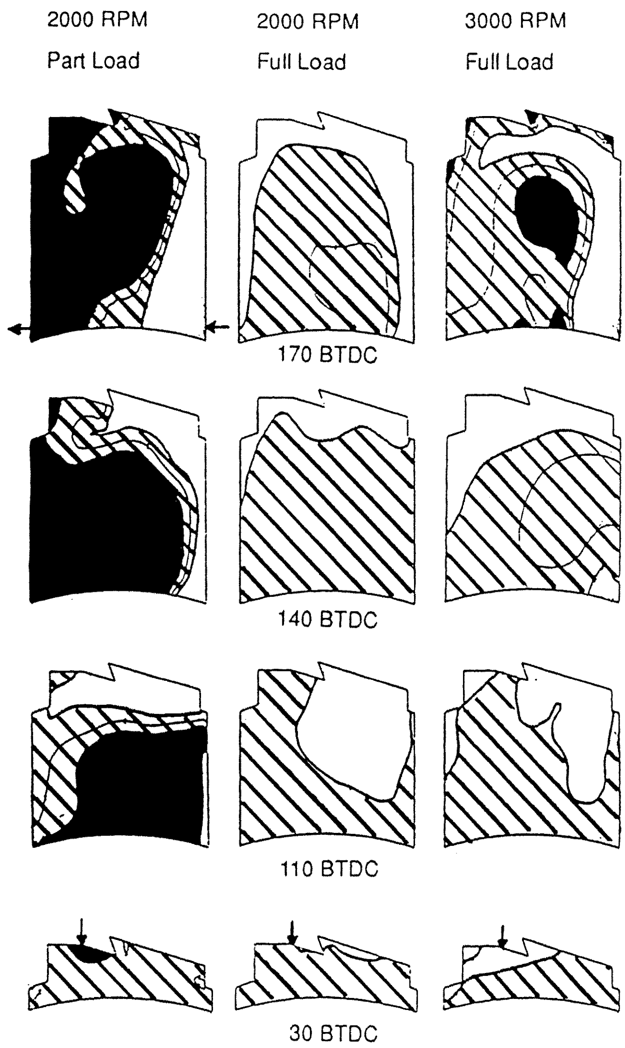

Fig. 7 Scavenging Loop Characterisation :

Contour-plots of Residual Gases Mass Fraction

(1) Spark plug

Less than 10%

10% to 30%

Over 30%

APPLICATION OF THE 3D CODE TO STUDY THE EFFECTS OF GEOMETRY CHANGES

In order to check the ability of the code to give a suitable account of geometry variations, we have compared two configurations which differ by the shape and position of the transfers and exhaust (figure 9). It therefore seemed us particularly important to refine our hypotheses relative to the orientation of the velocity vectors at the various orifices. We relied on the work of Blair (9) as well as on some hot wire anemometric measurements carried out on a fairly similar configuration and we took into account some horizontal and vertical deflections.

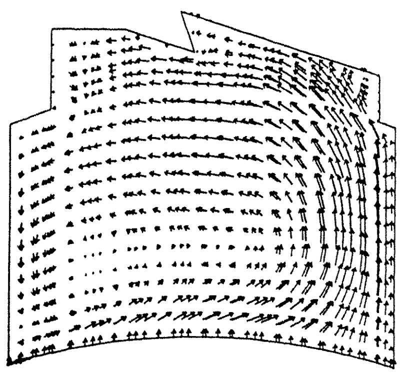

Fig.8 Fluid Movement at 90 BTDC Visualisation

2000 RPM Part Load

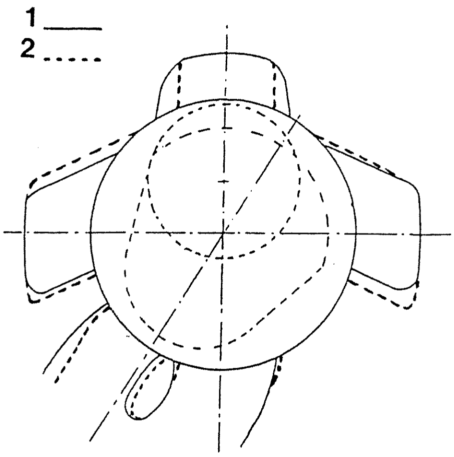

Fig. 9 Survey of the 2 Geometries

(1) Configuration No 1

(2) Configuration No 2

The only available experimental data had been obtained at 3000 RPM full load and the comparison was made at this operating point. Having carried out our treatment, the code supplies us with the following elements at the end of compression : in both cases, there is a rich mixture in the spark plug zone, but this is more diluted in the case of configuration No 1 (figure 10); these informations are consistent with our experimental observations which showed a greater stability of configuration No 2.

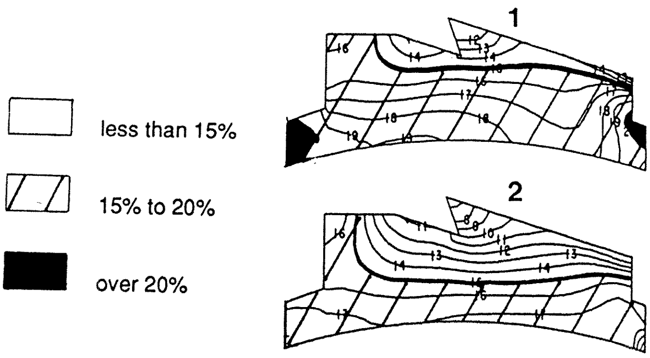

Fig. 10 Residual Gases Mass Fraction at 30 BTDC

(1) Configuration No 1

(2) Configuration No 2

CONCLUSION

In order to obtain the values of flow rates that are required for a good evaluation of IAPAC two-stroke engine's internal aerodynamics, we have combined engine test bench experiments and 0D numerical simulations; this way of doing already brings very interesting informations.

The work has then gone on with the help of the 3D code. In spite of some incertitudes bearing on velocity fields at the ports, the part load working analisys has been completed. Our goal which was to understand the results obtained at the engine test bench is reached; better still, a slightly more advanced analisys allows us to derive the way we must work to improve the engine's behaviour.

With the idea of numerically working the improvment of our engine - a way of doing which would allow us to reduce the number of engine test bench experiments -, we also wanted to see how the computer program takes into account some geometrical variations. Even if it is advisable to be more cautious about the interpretation of these last results (that may appear too fragmentary), we can say that they are encouraging as there is again an agreement between numerical and experimental tendancies.

NOMENCLATURE

k cell turbulence kinetic energy

u cell internal energy

ε cell turbulent kinetic energy dissipation rate

ρ cell density

ACKNOWLEGEMENTS

The authors wish to thank Mr Keribin, Moreau, Pahon and Souhaité for their help and many fruitful discussions.

REFERENCES

1 DURET, P., ECOMARD, A. and AUDINET, M., "A New Two-Stroke Engine with Compressed-Air Assisted Fuel Injection for High Efficiency Low Emissions Applications", SAE Paper No 880176, 1988.

2 DURET, P. and MOREAU, J.F., "Reduction of Pollutant Emissions on the IAPAC Two-Stroke Engine with Compressed Air Assisted Fuel Injection", SAE Paper No 900801, 1990.

3 HENRIOT, S., Le COZ, J.F. and PINCHON, P., "Three Dimensional Modeling of Flow and Turbulence in a Four-Valve Spark Ignition Engine - Comparison with LDV Measurements", SAE Paper No 890843, 1989.

4 PINCHON, P., "Three Dimensional Modeling of Combustion in a Prechamber Diesel Engine", SAE Paper No 890666, 1989.

5 AMSDEN, A.A, RAMSHAW, J.D., O'ROURKE,P.J. and DUCOWICZ, J.K., "KIVA : A Computer Program for Two- and Three-Dimensional Fluid Flows with Chemical Reactions and Fuel Sprays", LOS-ALAMOS Report LA-10245-MS, 1985.

6 AMSDEN, A.A, RAMSHAW, J.D., CLOUTMAN, L.D. and O'ROURKE,P.J., "Improvements and Extensions to the KIVA Computer Program", LOS-ALAMOS Report LA-10534-MS, 1985.

7 REYNOLDS, W.C., "Modeling of Fluid Motions in Engines - An Introductory Overview", Combustion Modeling in Reciprocating Engines, Plenum Press, New York, 1980.

8 DIWAKAR, R. and EL TAHRY, S.H., "Comparison of Computed Flow-fields and Wall Heat Fluxes with Measurements from Motored Reciprocating Engine-like Geometries", Third International Computer Engineering Conference, CHICAGO, 1983.

9 SMYTH, J.G., KENNY, R.G. and BLAIR, G.P., "Steady Flow Analysis of the Scavenging Process in a Loop-Scavenged Two-Stroke Cycle Engine - A Theoretical and Experimental Study", SAE Paper No 881267, 1988.