Asociación Argentina de Mecánica Computacional

Mecánica Computacional Vol XXXIII, págs. 2541-2553 (artículo completo)
Graciela Bertolino, Mariano Cantero, Mario Storti y Federico Teruel (Eds.)
San Carlos de Bariloche, 23-26 Setiembre 2014

COMPUTATIONAL SIMULATION FOR AN INTERNAL COMBUSTION ENGINE WITH TORCH IGNITION SYSTEM

Fabrício T. Borghi[a], Marco T. de Faria[b] e Ramon M. Valle[c]

[a]*CTM Centro de Tecnologia da Mobilidade, Universidade Federal de Minas Gerais, 31270-901,Minas Gerais, MG, Brasil, fabricio.tb@gmail.com, http://www.demec.ufmg.br/ctm*

[b]*CPH Centro de Pesquisas Hidráulicas e Recursos Hídricos, Universidade Federal de Minas Gerais, 31270-901,Minas Gerais, MG, Brasil, mtfaria@demec.ufmg.br, http://www.demec.ufmg.br/cph*

[c]*CTM Centro de Tecnologia da Mobilidade, Universidade Federal de Minas Gerais, 31270-901,Minas Gerais, MG, Brasil, ramon@demec.ufmg.br, http://www.demec.ufmg.br/ctm*

Keywords: CFD, IC Engine, Prechamber, Torch, Ignition.

Abstract. Stringent automotive emissions and fuel economy regulations have been bringing challenges for the development of new engine technologies to achieve greater levels of efficiency and pollutants reduction. In this scenario the homogeneous charge pre-chamber jet ignition system (HCJI) enables lean operation due the jet combustion gases emerging from the small pre-chamber combustor as the ignition source for main chamber combustion in an internal combustion engine. The present computational work was carrying out to investigate the interaction between the pre-chamber and main chamber fluid dynamics events. This CFD research was performed and validated with a experimental data for a single cylinder of a 4-stroke indirect fuel injection engine under the motoring condition running at 4500 rpm with 50% wide open throttle condition. The analyses are focused to identify the influences in the fluid flow field in the terms of tumble ratio and the turbulence kinetic energy along the degree of crank angle during intake and compression stroke. The computational results can explain better the homogeneity with this new combustion system configuration.

1 INTRODUCTION

During the past 20 years, significant advances in controlling exhaust emissions from vehicles have been made where the emissions reductions were achieved through readjustment and control of engine operating conditions (Beckman et al., 1967). In the literature, highly effective exhaust treatment devices based on thermal and catalytic oxidation of hydrocarbons and carbon monoxide has been applied in the engine exhaust system (Cantwell et al., 1966; Bartholomew, 1966; Campion et al., 1972). On the other hand, basic combustion process modification as an alternative means for emissions control. The lean combustion process in SI engines is one of this alternative and it has been demonstrated a significant pollutant reductions without need for exhaust treatment devices external to the engine. This process was studied first during 1908 to demonstrate the advantages of higher thermal efficiency (Hopkinson, 1908; Jakson, 1964). In addition the lean combustion technology shows to offer the lower emissions and also improves the fuel economy (Tanuma et al., 1971; Ricardo, 1923-24).

Charge stratification technique within an internal combustion engines operating with lean fuel-air ratios has been achieved by involving the ignition of a very small and localized quantity of fuel-rich mixture, which in turn serves to inflame a much larger quantity of surrounding fuel-air mixture too lean for ignition under normal circumstances. This process correspond a reduced exhaust emissions and fuel consumption. Among all the researches related this technique, Ford PROCO (Programmed Combustion Process) is a direct injection stratified charge engine which is dependent upon a combination of swirl and injection timing to facilitate the charge stratification process (Kowalewicz, 1984; Haslett et al., 1976; Newhall, 1975). The Texaco Controlled Combustion System (TCCS) engine is another direct injection stratified charge engine which is dependent upon tile air swirl to provide mixture formation. (Haslett et al., 1976; Newhall, 1975; Alperstein et al., 1974) The design is similar to that of the Ford PROCO, but instead of varying the injection timing the injection rate alone is varied to give the required power.

Pre-chamber combustion system consists in the division of the combustion region into two adjacent chambers. This pre-chamber is a small ignition chamber equipped with a spark plug that communicates with the much larger main combustion chamber connected by a nozzle. The pre-chamber is filled with fuel-rich fuel-air mixture while a very lean and normally unignitable mixture is supplied to the main chamber. Due the expansion of high temperature flame products from the prechamber leads to ignition and burning of the lean main chamber fuel-air charge. The first use in igniting lean mixtures was under taken by Ricardo (1922) indicated that the engine could perform very efficiently within a limited range of controlled operating conditions. Their commercial applications were not realized as conventional engines became cheaper to manufacture, and developments within the conventional engine reduced the need for stratified charge engines.

The literature show us that many researches and automotive companies already have been conducting works related with this kind of technology as the case of the Porsche SKS that has a pre-chamber engine that utilizes fuel injection into both pre-chamber and main chamber for mixture formation (Gruden, 1976). The main mixture injection occurs in the inlet manifold whereas the pre-chamber injection is direct. The VW Pre-Chamber Injection System (Pischinger and Klocker, 1974; Kuck and Brandstetter, 1975; Brandstetter, 1976) is similar to the Porsche in that fuel is directly injected to the pre-chamber and the main charge is formed externally. However where the Porsche system used turbulence to increase the burn rate, the VW system uses swirl generated by the incoming charge from the main chamber which passes through an offset connecting passage. The Honda Compound Vortex Controlled

Combustion (CVCC) system (Date et al., 1974; Yagi et al., 1980; Gussak et al., 1975) uses a secondary carburetor to supply a relatively rich mixture to the pre-chamber through a secondary inlet valve. Its operation is based upon a three zone stratification system, whereby during the induction stroke a proportion of the rich mixture leaves the pre-chamber and mixes with a proportion of the lean charge in the main combustion chamber. During compression this intermediate charge is compressed into the connecting passage between the pre-chamber and the main chamber.

Other factors have been studied for pre-chamber systems. Wyzczalek et al. (1975) used electronic fuel injection into a pre-chamber. Gussak et al. (1979) showed that the use of a nozzle connecting the prechamber to the main chamber promoted noise generation from the engine. Ryu et al. (1987) studied the effect of torch jet direction on the performance of a pre-chamber stratified charge engine. Purins (1974) showed that the air/fuel ratio of the pre-chamber at ignition was a function of the of the angle of ignition, i.e as the ignition angle was retarded more of the lean mixture was forced into the pre-chamber due to the compression action of the piston, hence increasing the proportion of air in the pre-chamber. Pischinger and Adams (1980) found that by increasing the cylinder charge motion the fuel consumption was improved under part load, as well as reducing the emissions because more complete combustion was obtained.

The current study was carried out as part of a larger research and project development involving computational and experimental aspects, describing an innovative combustion system that can be applied for all types of internal combustion engines. The motivation of this work was driven by the necessity to develop innovated engines that meet heavy governmental legislations requirements. The computational and experimental development for an engine with Torch Jet ignition has been developed through financial investment from Petrobras S.A. In this work an overall computational analysis will be show for Torch Jet Ignition, detailing the cold flow and the parameters that influence this kind of system. The computational simulation results were validated with an experimental data.

During recent years, besides experiment on the engine dynamometer, researches applied for in-cylinder flow involving numerical modeling has been intensively by the use of computational tools (Rakopoulos et al., 2010). Lim and Min (2005) studied the optimization of combustion chamber geometry along with swirl and selected spray parameter is the key to reducing pollutant emissions and for better fuel economy. Stephenson et al. (1996) conducted a parametric study on the effects of swirl, initial turbulence, oxygen concentration and ignition delay on fuel vaporization, mixing and combustion process. The effects of intake flow field was further investigated by varying the geometry of the intake ports and intake runners in a model dual port direct injection diesel engine in KIVA simulations. Payri et al. (2004) have carried out the CFD modeling of the in-cylinder flow in direct-injection Diesel engines for the intake and compression stroke with different combustion chambers and validate their numerical result with the experimental work Payri et al. (2004) and Auriemma et al. (1998) made in-cylinder measurements under motoring conditions in a light duty diesel engine equipped with a re-entrant bowl in piston combustion chamber. Prasad et al. (2011) observed the effect of piston bowl for swirl and turbulent kinetic energy intensification around top dead center through computational simulations. Chen et al. (1998) simulated the transient flow for the induction stroke using STAR-CD software and observed that the standard k-ε model with wall function for description of boundary layer behavior predict the fully turbulent flow inside the cylinder.

Jayashankara and Ganesan (2010) carried out the CFD investigation to study the effect of fuel injection timing and intake pressure on the performance of a direct injection in diesel engine. Hyun et al. (2002) has performed the numerical simulation by employing the CFD code of KIVA-3 whereby the shape of combustion chamber, swirl intensity, and injection timing are modified and the effect of mixture formation is investigated. Bianchi et al. (2003) investigated the influence of different initial condition procedures on combustion and emissions predictions in small-bore high-speed direct injection diesel engine. The analysis was performed by using the STAR-CD code for the intake stroke calculation and KIVA for the compression stroke and combustion simulation. Kim et al. (1999) detailed the in-cylinder air motion during the intake and compression stroke to examine the interaction of air motion with fuel spray injected directly into the cylinder.

Based on literature few computational works has been found about torch jet ignition using pre-chambers. Charlton et al. (1990) conducted a computational research based on the stratification of combustible gas charges using the sectional combustion chamber. In this research the numerical modelling of the mixture preparation process used PHOENICS code where the results were validated through experimental results. After Roethlisberger and Favrat (2003) investigated combustion system with a pre-chamber to stationary biogas fuelled engines. They applied the KIVA-3V code to calculate and further to optimize pre-chamber shape and find best working conditions for the 6-in-line, turbocharged, intercooled, heavy duty engine. The studies of Roethlisberger and Favrat (2003) on improving the combustion process by moving the ignition point from the main combustion chamber of conventional engine to the small prechamber were carried out in the Swiss Institute of Technology in Lausanne. The experimental researches were the continuation of numerical studies performed in KIVA-3V code.

In the present work numerical and experimental techniques were applied in order to study the in-cylinder flow field in a commercial four valve spark ignition engine with and without the pre-chamber. Investigation was focused in analyze the generation and evolution of tumble-vortex structures during the intake and compression strokes, and the capacity of this engine to promotes the turbulence enhancement during tumble-vortex degradation, at the end of the compression stroke.

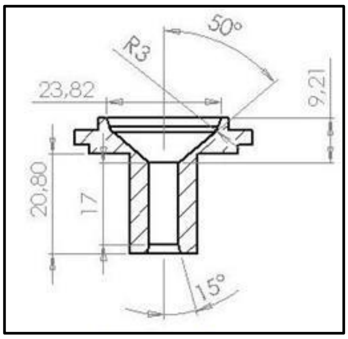
2 PRE-CHAMBER ENGINE TECHNOLOGY

To ignite very lean charge is very difficult and it burns so slowly that is necessary a special ignition source than a conventional spark plug. A pre-chamber in an Otto engine works as a powerful ignition source and this ignition occurs almost simultaneously over a larger volume in the main combustion chamber. The conventional pre-chamber consists of a small volume with a fuel injector that is responsible to filled it with a very rich charge or in some cases with pure fuel in the gas phase, during the intake stroke. During the compression stroke the gas present in the pre-chamber is quick mixed with the poor charge provided by the intake system. The combustion is started in the pre-chamber and the combustion rises the temperature and pressure inside the pre-chamber. This higher pressure generates a jet of burned gas to flow into the main chamber and ignite the lean main charge. Nozzles are responsible by the connection between the pre-chamber and main chamber. The nozzles change the direction of the flow during the expansion stroke.

Turkish (1975) worked with the first pre-chambers used for the Otto cycle that were adaptations of diesel pre-chamber engine with spherical shape. Wolff et al. (1997) performed experiments with cylindrical and conical geometries of pre- chambers and compared their performances. It was concluded that pre-chambers of cylindrical shape were better because they tend to form more homogeneous mixtures inside, due to the large vortex formation.

Although several pre-chambers have been designed to work with stratified mixture, that means, the fuel is injected into the pre-chamber too, the torch ignition system to be studied in this work involves only a pre-chamber geometry with homogeneous mixing in a typical four-stroke engine with port injection system. The pre-chamber is positioned in the place of spark plug, position at the vicinity area of the engine where this region provides the homogeneous mixture for a rapid combustion process.

The proposed pre-chamber consists of a small geometry coupled by a divergent nozzle directed connected across the main combustion chamber in the same position of the spark plug. The Figure 1 represents the proposed configuration and your respective assembly in the internal combustion engine. Therefore, the flow of gases in combustion is going directly to the encounter of the top of the piston, generating larger turbulence (Ryu et al., 1987; Turkish 1975).



(a)

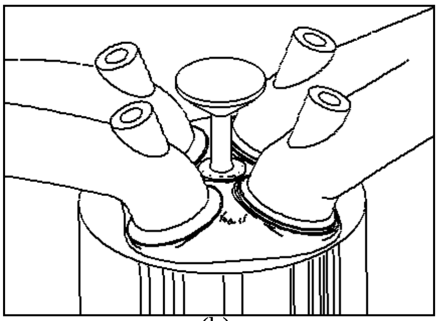
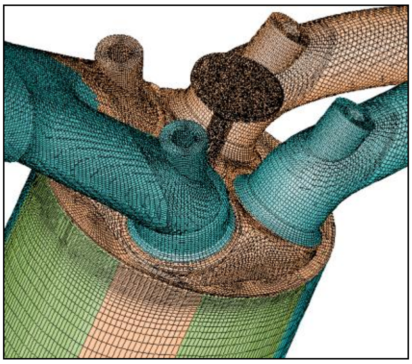

(b)

Figure 1: Proposed pre-chamber shape (a) and system assembly (b).

3 MESH MODELING

The computational software ES-ICE was responsible for the grid generation and moving mesh through automatic dynamic addition and removal of cells, which facilitates control over mesh resolution and distortion in moving-boundary problems. The generated meshes consist of hexahedral cells for the engine model including the intake and exhaust ports. With the objective of achieve a better accuracy and stability for the CFD calculations, the number of hexahedral cells has been adopted varies from 300,000 cells in TDC and around 700,000 cells in BDC, where a fine grid is necessary for mesh snapping during the valve movement.

For the pre-chamber a fine hexahedral grid is necessary to capture all the flow instabilities and turbulence field. The number of fixed cells has been adopted is from 500,000 cells. This approach is used to obtain a good quality mesh, especially for the connectivity between pre-chamber and the main chamber. The Figure 2 represents the computational mesh for the assembly and vales.


(a)

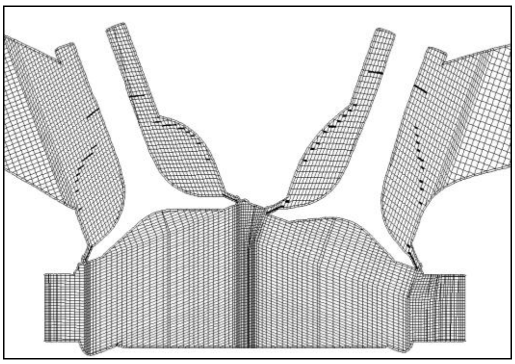

(b)

Figure 2: Computational mesh representation for the engine (a) and the valves assembly (b).

4 NUMERICAL METHODOLOGIES

To evaluate a compressible and transient fluid flow simulation for the proposed torch ignition system, the commercial code STAR-CD for finite volume method has been used to solve the discretized continuity, energy and Navier-Stokes equations. The numerical methodology in this work is based on the PISO algorithm for pressure-velocity coupling and the second upwind differencing scheme (MARS) as the spatial discretization is used for the momentum, energy and turbulence equations. The RNG k-ε turbulence model was chosen for this application. The implicit method for temporal discretization with a variable time step depending on the stage of the engine cycle was chosen due the high local velocities in the valves and nozzle zones.

The cold flow simulation is started when intake valve opening and continued to the compression stroke without the fuel injection. The initial and boundary conditions values for pressure and temperature on inlet and outlet boundaries, at engine operating speed of 4500 rpm, were obtained from the experimental measurements in engine dynamometer and these variables were considered as homogeneous in the whole domain. The initial in-cylinder turbulent intensity is set at 3 % of the mean flow, and the integral length scale is set at 0.1 m. The fixed temperature boundary conditions were specified independently for the cylinder head, the cylinder wall, and the piston crown as 450 K, 400 K and 500 K. The adiabatic condition was considered for the walls of the intake and exhaust ports including the lateral walls of the valves.

5 COMPUTATIONAL VALIDATION

The experimental work is developed for the purpose of validating the computational fluid dynamics (CFD) simulation of a real engine. The available data used for the computational validation was performed by considering the operation of one cylinder without combustion. The comparison was performed in the engine speed of 4500 rpm and the intake port temperature and pressure are 299 K and 0.98 bar. There are great agreements for the comparison between the calculated and experimental pressure curves during intake and compression stroke for both engines with and without pre-chamber. It is indicated that the calculated maximum pressure is in a close agreement with the measured result. The Figure 3 shows the agreement between the experimental and computational work for pressure curve validation.

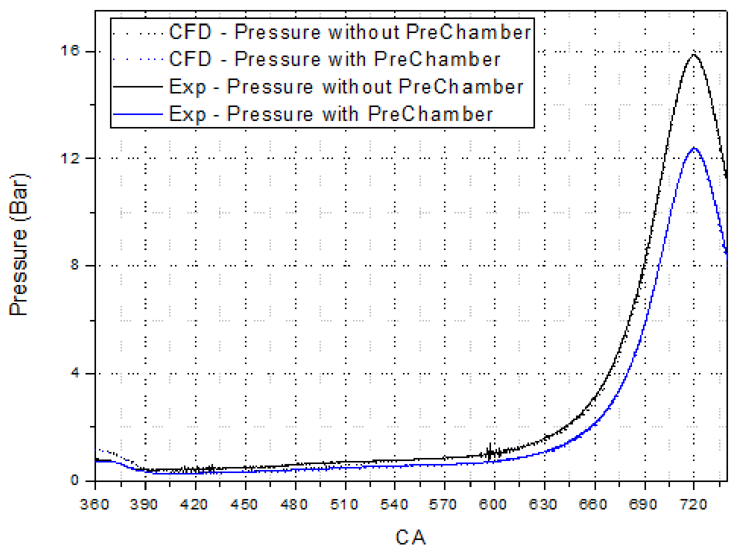


Figure 3: Computational validation.

6 RESULTS

The numerical study has been carried out to identify which influences the pre-chamber has in the fluid flow field and turbulence profile for the present engine without combustion. For this purpose, tumble ratio for both systems on the transversal and normal directions is calculated for every crank angle degrees of engine cycle. This dimensionless parameter is generated during intake and compression stroke due to the high turbulence in the cylinder and they are applied to quantify the rotational and angular momentum motion inside the cylinder.

The tumble ratio (m/s) is defined by the Equation 1 where X_i and Z_i are the centroid coordinate of cell i (m), X_m and Z_m are the centre of mass of cylinder (m) and the u_i and w_i are the velocity at cell i (m/s).

$$\mathrm{TumbleY} = \frac{\sum_{\mathrm{Cells}} \rho_i . V_i \left[(Z_i - Z_m). u_i - (X_i - X_m). w_i \right]}{\sum_{\mathrm{Cells}} \rho_i . V_i \sqrt{(X_i - X_m)^2 + (Z_i - Z_m)^2}} \tag{1}$$

It was considered a velocity field during the intake stroke at 60º after top dead center (TDC) with a intake valve lift of 6 mm on the XZ cutting plane at Y = 0 mm as shown in Figure 4 and Figure 5. The flow field for the engine without the pre-chamber is characterized by two median vortexes, one at each sideway generated by the intake valves. These structures have different rotating direction, so both the vortex favours the flow entrance generated by valve zone. It can be seen that there are strong jet flows in the zone near the valve curtains because there is a medium flow and velocity field in this crank angle and small vortex spins create instabilities along the jet flow on both sides. In the same figure can be seen that a toroidal vortex is developed in the centre of the dome due the jet motion. Analyzing the engine with the pre-chamber it can be seen that the toroidal vortex is almost dissipated and the pre-chamber is just starting to be filled during the intake stroke. The pre-chamber’s filling decreases the jet flow in the middle of the engine cylinder, consequently reducing the sideway vortex and the small vortex spins are completed dissipated in the flow field.

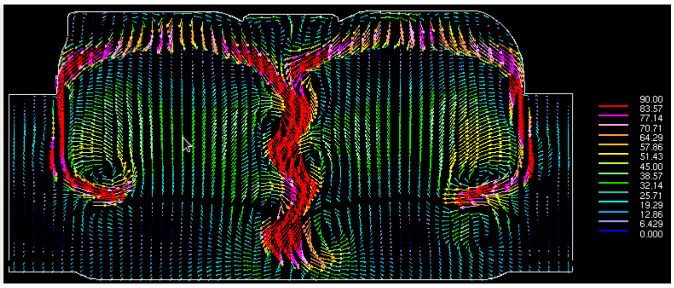


Figure 4: Velocity profile of engine without pre-chamber for 6 mm of intake valve lift

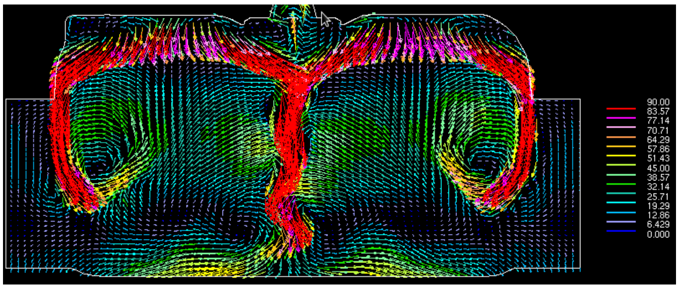


Figure 5: Velocity profile of engine with pre-chamber for 6 mm of intake valve lift

Figure 6 and Figure 7 shows the intake stroke at 110º after top dead center (TDC) with is the maximum intake valve lift of 8.5 mm at the same XZ. This flow field for the engine without the pre-chamber differs for the last one by two big vortex, one at each side of the jet flow and these structures have different rotating direction. The toroidal vortex and the pre-chamber's filling follow the same behaviour of the previous analysis. The flow field for the engine with the pre-chamber is characterized by four median vortexes, two at each side generated by the intake valves and these structures have different rotating direction. Both vortexes improve the homogeneity of the flow. It can be seen that there are symmetry in the flow pattern under both valve regions.

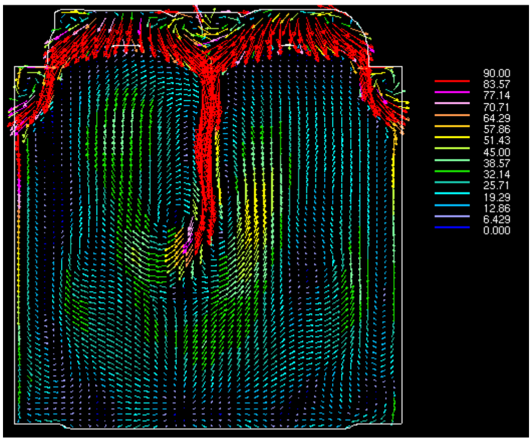


Figure 6: Velocity profile of engine without pre-chamber for 8.5 mm of intake valve lift

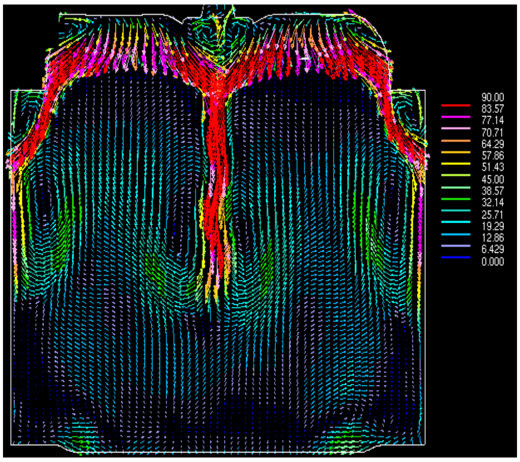


Figure 7: Velocity profile of engine with pre-chamber for 8.5 mm of intake valve lift

The investigation of tumble ratio variation inside engine cylinder in the normal direction can be figured out in Figure 8. Tumble ratio has a negative value due to its rotating direction and it starts to change its magnitude at the early part of intake stroke. By 130º after top dead center (TDC), a dominant tumbling motion has been developed and decreased incrementally as the valve lift closes until the end part of the compression stroke. It can be concluded that the engine with pre-chamber can generate lower tumble ratio on the normal side during the intake valve open corresponding the lifts between 3 to 8.5 mm and in the period of 130º after top dead center (TDC) in the admission stroke until approach bottom dead center (BDC).

The turbulence kinetic energy is a measure of the characteristics speed of the turbulent flow over a distance characteristic of the flow structure. The production of turbulence kinetic energy is very important in characterizing the in-cylinder locations where turbulence is being created. The in-cylinder turbulence can be beneficial for gaining the optimum air–fuel mixing preparation before fuel injection and increasing combustion rates.

During the early stages of the intake stroke, the jet flow interactions are the most important mechanism for the production of turbulence. The early fluctuation of turbulence intensity is due to the sudden opening of intake valve and closing of exhaust valve. In the Figure 9, after 140º of top dead center (TDC) for the engine without prechamber, the turbulence intensity increases rapidly and reaches its maximum value in the intake stroke. However the prechamber system reaches its maximum value after 90° of top dead center (TDC).

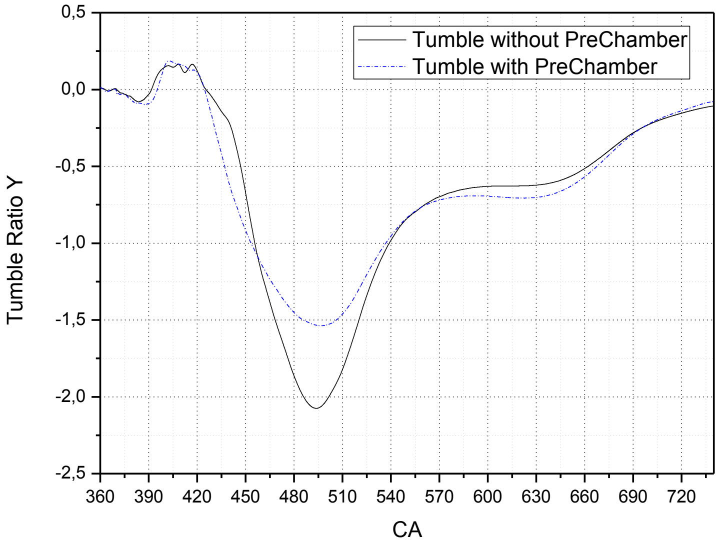


Figure 8: Tumble ratio

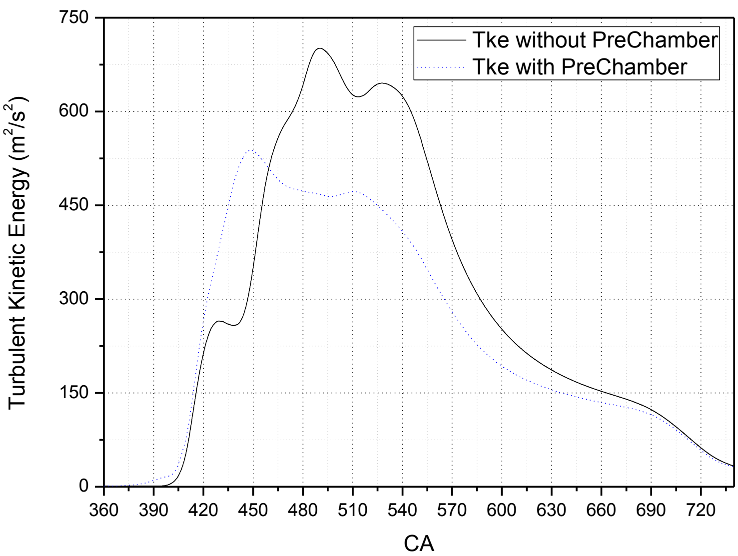


Figure 9: Turbulent kinetic energy.

The turbulence kinetic energy are showed in the Figure 10 and Figure 11, and for the maximum valve lift, the turbulence production associated with the jet interaction penetrates to the middle of the engine cylinder has higher magnitude for the engine configuration without prechamber than the engine configuration with prechamber. Comparing both the engine configurations is possible to verify that the engine configuration with prechamber gives a homogeneous turbulent kinetic energy profile.

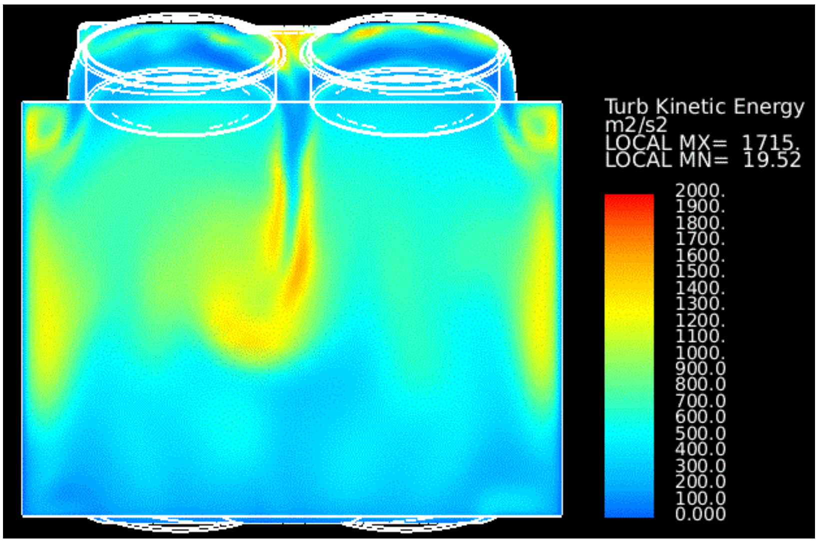


Figure 10: Turbulence kinetic energy profile of engine without pre-chamber for 8.5 mm of intake valve lift

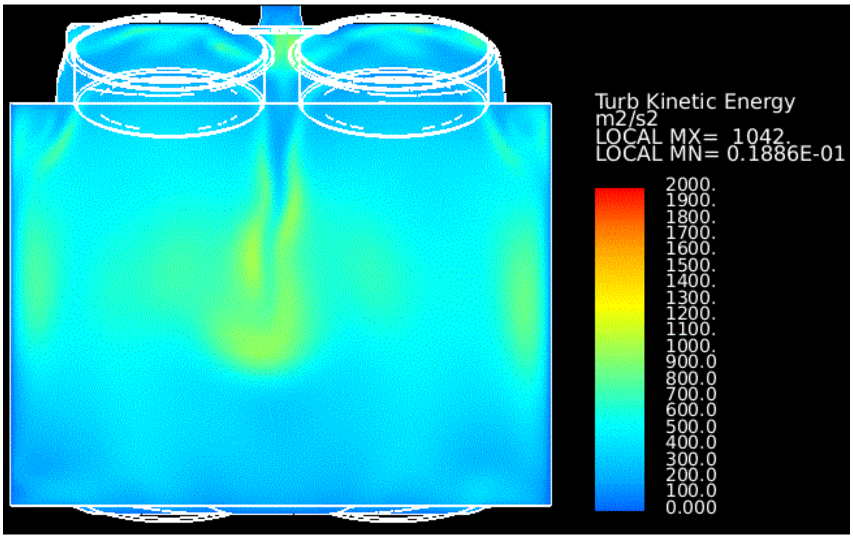


Figure 11: Turbulence kinetic energy profile of engine with pre-chamber for 8.5 mm of intake valve lift

7 CONCLUSIONS

The computation solved for the transient analysis of intake and compression strokes for motored engines with and without the torch ignition system were carrying out for engine speed at 4500 rpm, having a good agreement with the experimental tests. In this work, a proposed geometry model for the pre-chamber was developed and the intake calculations involving moving valves and piston, possibility that the flow field could be analyzed completely, showing the in-cylinder airflow influence of the prechamber under the main combustion chamber.

The fluid flow and turbulence characteristics acquired from the simulation were taken into account to verify the better homogeneity of air structures of prechamber engine compared with the engine without prechamber. This homogeneity, in the beginning, could generates a smaller air-fuel mixture process inside the main cylinder but it is possible to observe that the turbulent kinetic energy increases more quickly during the intake valve open for the engine with prechamber giving a better air-mixture process. By the other hand, the mixture process also are due by the tumbling phenomenon and for this reason the air-fuel mixture for the engine with prechamber can keep an adequate mixture process for two different stages, one during the admission stroke until the maximum intake valve open and the other during the compression stroke period.

The prechamber air filling is strongly influenced by your position in the main combustion chamber and the geometry design. The results has been proven that for a homogeneous engine charging the prechamber air filling starts since the intake valve open and increase during the compression stroke. The prechamber filling stage is very important during the intake valve open because in this period the turbulent kinetic energy magnitude is higher, providing a better air-fuel filling that will leading a better combustion process.

ACKNOWLEDGMENTS

The authors would like to thank the *Petrobras S.A.* for sponsoring this work.

REFERENCES

Alperstein, M., Schafer, G.H. and Villforth, F.J., "Texacos stratified charge engine - multifuel, efficient, clean and practical", SAE 740563, 1974.

Auriemma, M., Corcione, F. E., Macchioni, R. and Valentino, G., “Interpretation of air motion in reentrant bowl in-piston engine by estimating Reynolds stresses” SAE Paper No. 980482, 1998.

Bartholomew, E., Society of Automotive Engineers, Paper 660109, 1966.

Brandstetter, W.R., “Experimental results from a Volkswagen's pre-chamber stratified charge engines”, C249/76, IMechE, 1976.

Beckman, E. W., Fagley, V. S., and Sarto, J. O., Society of Automotive Engineers Transactions, Vol. 75 (1967).

Bianchi, G.M., and Fontanesi, S., “On the applications of low-Reynolds cubic k-ε turbulence models in 3D simulations of ice intake flows”, SAE Paper No. 2003-01-0003.

Campion, R. M., et al, Society of Automotive Engineers, Publication SP-370, 1972.

Cantwell, E. N., and Pahnke, A. J., Society of Automotive Engineers Transactions, Vol. 74 (1966).

Charlton, S.J., Jager, D.J., Wilson, M., “Computer Modelling and Experimental Investigation of a Lean Burn Natural Gas Engine”, SAE Paper 900228, 1990.

Chen, A., Veshagh, A. and Wallace, S., “Intake flow predictions of a transparent direct injection diesel engine”, SAE Paper No. 981020, 1998.

Date, T., Yagi, S., Ishizuva, A. and Fujii, I., "Research and development of the Honda CVCC engine." SAE 740605, 1974.

Gruden, D., Markovac, U. and Lorcher, H., “Development of the Porsche SKS engine”, IMechE, 1976.

Gussak, L. A., Turkish, M., and Siegla, D., "High Chemical Activity of Incomplete

Combustion Products and a Method of Prechamber Torch Ignition for Avalanche Activation of Combustion in Internal Combustion Engines," SAE Technical Paper 750890, 1975.

Gussak, L.A., Karpov, V.P. and Tikhnov Y.V., “The application of LAG-Process in pre-chamber engines." SAE 790692, 1979.

Haslett, R.A., Monaghan, M.L. and McFadden, J.J., “Stratified charge engines" SAE 760755, 1976.

Hopkinson, B., “The effect of mixture strength and scavenging on thermal efficiency”, Proc. Inst. Mech. Engr., 417, 1908.

Hyun, G., Oguma, M. and Goto, S., “3-D CFD analysis of the mixture formation processes in an LPG DI SI engine for heavy duty vehicles.” Proceedings of the 12th International Conference of Multidimensional Engine Modelling User Group Meeting SAE Congress, Detroit, Michigan, USA, 2002.

Jakson, M.W., “The influence of air-fuel ratio, spark timing, and combustion chamber deposits on exhaust hydrocarbon emissions”, SAE technical progress series, Vehicle emission Vol. 6, pp. 175-194 ,1964.

Jayashankara, B. and Ganesan, V., “Effect of fuel injection timing and intake pressure on the performance of a DI diesel engine – A parametric study using CFD”, Energy Conversion and Management vol (51), 2010 pp 1835–1848.

Kim, Y.J., Sang, H.L., and Nam, H.C., “Effect of air motion on fuel spray characteristics in a gasoline direct injection engine”, SAE Technical Paper Series No. 1999-01-0177, 1999.

Kowalewicz, A., "Combustion systems of high speed piston LC engines" Elsevier science publishing Co., 1984.

Kuck, H.A. and Brandstetter, W.R., “Investigations on a single cylinder stratified charge engine with a scavenged pre-chamber”, C92/75, IMechE, 1975.

Lim, J. and Min, K., "The Effects of Spray Angle and Piston Bowl Shape on Diesel Engine Soot Emissions Using 3-D CFD Simulation," SAE Technical Paper 2005-01-2117, 2005.

Newhall, H.K, "Combustion process fundamentals and combustion chamber design for low emissions." SAE 751001, 1975.

Payri, F., Benajes, J., Margeo, X., Gil, A ., “CFD modeling of the in-cylinder flow in direct injection diesel engine”, Computers & Fluids; vol(33), 2004, p. 995–1021.

Pischinger, F. F. and Klocker, K.J., "Single-cylinder study of stratified charge process with pre-chamber injection." SAE 741162, 1974.

Pischinger, F. and Adams, F., "Influence of intake swirl on the characteristics of a stratified charge engine with pre-chamber injection" C391/80, IMechE, 1980.

Purins, E.A., "Pre-chamber stratified charge engine combustion studies", SAE 741159, 1974.

Prasad, B.V.V.S.U., Sharma, C.S., Anand, T.N.C. and Ravikrishna, R.V., “High swirl-inducing piston bowls in small diesel engines for emission reduction”, Applied Energy, vol(88), 2011, p2355–2367.

Rakopoulos, C. D., Kosmadakis, G.M. and Pariotis, E.G., “Investigation of piston bowl geometry and speed effects in a motored HSDI diesel engine using a CFD against a quasi-dimensional model”, Energy Conversion and Management; vol (51),3, 2010, pp. 470–84.

Ricardo, H.R., “Report of the empire motor fuel committee”, Procinst. autoengg .Part 1, 1923-24 .

Ricardo, H. R., SAE Journal, Vol. 10, pp. 305-336, 1922.

Roethlisberger, R.P. and Favrat, D., “Investigation of the prechamber geometrical configuration of a natural gas spark ignition engine for cogeneration: part I, numerical simulation”, Int. J. Therm. Sci. 42 (3) (2003) 223–237.

Roethlisberger, R. P. and Favrat, D., “Investigation of the prechamber geometrical configuration of a natural gas spark ignition engine for cogeneration: part II, experimentation”, Int. J. Therm. Sci. 42 (3) (2003) 239–253.

Ryu, H.; Chtsu, A. and Asanuma, T., “Effect or torch jet direction on combustion and performance of a prechamber spark-ignition engine”, Paper SAE 870167, 1987.

Stephenson, P.W., Claybaker, P.J. and Rutland, C.T., “Modeling the effects of intake generated turbulence and resolved flow structures on combustion in direct injection diesel engine”, SAE Paper No. 960634, 1996.

Tanuma, T., Saski K., Kaneko,T. and Kawasaki, H , “Ignition, combustion ,and exhaust emissions of lean mixtures in automotive spark ignition engines”, SAE paper 710159,1971.

Turkish, M. C., “Prechamber and valve gear design for 3-valve stratified charge engines”, Paper SAE 751004, 1975.

Wolff, D., Tamura, M., Tai, H and Sakurai, T., “Looking into the prechamber of a lean-burn gas engine”. JSME International Journal, vol. 40, N° 2, 1997.

Yagi, S., Fujii, I., Ajiki, V. and Tsuada, T., "The anti-knock quality in the stratified charge engine with an auxiliary combustion chamber", C395/80, IMechE, 1980.