Comparative study of the indicated cycle of a diesel engine using simulation CFD and experimental data

Estudio comparativo del ciclo indicado de un motor diesel mediante simulación CFD y datos experimentales

ALVARO L. DELGADO-MEJIA
Magister en Ingeniería: Energética
Institución Universitaria Pascual Bravo
alvaro.delgado@pascualbravo.edu.co
Medellín, Colombia

LUIS C. OLMOS-VILLALBA
Magister en Ingeniería: Energética
Institución Universitaria Pascual Bravo
luis.olmos@pascualbravo.edu.co
Medellín, Colombia

SAUL E. RIVERO-MEJIA
Magister (c) en Ingeniería: Infraestructura y Sistema de Transporte
Institución Universitaria Pascual Bravo
saul.rivero@pascualbravo.edu.co
Medellín, Colombia

Fecha de recibido: 02/10/2013
Fecha de aceptado: 26/03/2014

Forma de citar: DELGADO, Alvaro, OLMOS, Luis y RIVERO, Saul.Comparative study of the indicated cycle of a Diesel engine using simulation and experimental data. Rev.UIS.Ingenierías, 2014, vol.13, n.1, p.p 23-31.

ABSTRACT

In this paper, a comparative study between numerical simulations and experimental data of the indicated cycle of a direct injection diesel engine is presented. A CFD package and a predictive model were used to simulate several engine operating conditions. Results were compared with experimental data obtained from an engine test bench. The comparison was based on indicated-cycle parameters such as pressure, temperature, heat release, power, efficiency, specific fuel consumption and mean effective pressure. Results show that in all cases simulated cylinder-pressure curves are in very good agreement with the experimental results. For the low-load mode, differences are around 5% at maximum pressure peak. On the other hand, temperature and heat release rate curves present significant differences between simulations and experiments. This could be a consequence of the combustion and heat transfer models used for the simulations. However, it is shown that the use of CFD tools for studying combustion phenomena in diesel engines is highly convenient.

KEYWORDS: CFD solver, indicated cycle, Diesel Engine, predictive model, heat transfer.

RESUMEN

En este trabajo se presenta un estudio comparativo entre la simulación numérica y datos experimentales del ciclo indicado de un motor diesel de inyección directa. Para esto se han efectuado una simulación multidimensional utilizando un paquete CFD y empleando un modelo predictivo, para varias condiciones de operación del motor. Ambos resultados se han comparado con datos experimentales medidos sobre un motor montado en banco de ensayos. El estudio se ha hecho en términos de los parámetros habituales del ciclo indicado como la evolución temporal de presión, temperatura y calor liberado, así como de la potencia, eficiencia, consumo específico de combustible y presión media indicada.

Los resultados muestran que las curvas de presiones simuladas son bastante próximas a las medidas en todos los casos analizados, con las mayores diferencias en los modos de baja carga en torno a un 5% como mucho en el pico de máxima presión, aunque si se encuentran diferencias en las curvas de temperatura y de tasa de calor liberado, debido en parte a la calidad de los modelos de combustión y transferencia de calor empleados en la simulación. No obstante, queda demostrada la conveniencia del uso de herramientas basadas en CFD aplicadas al estudio de la combustión en motores.

Palabras claves: CFD, Ciclo indicado, Motor diésel, modelo predictivo, transferencia de calor.

1. INTRODUCTION

Nowadays environmental pollution from combustion engines is becoming a serious challenge for automobile manufacturers, because of the restrictive normative. In the case of diesel engines, efforts have been focused on increasing efficiency and decreasing NOx and particulate emissions. In order to achieve that, Computational Fluid Dynamics (CFD) modeling has been used as a tool for the simulation of internal combustion engines, with very good results that can be found in the scientific literature.

Diesel combustion is a very complex phenomenon, due to the amount of processes that take place simultaneously and because they depend on each other. In terms of CFD simulation, there is a compressible and transient flow, involving phase change, heat transfer by convection, conduction and radiation, premixed and diffusive combustion dominated by chemical kinetic, formation of contaminant species, etc., which are developed in a volume of variable geometry (Merker, A., et al 2004). The method is based on the spatial discretization of the system domain in small control volumes where the conservation laws are applied and solved for each time step. The disadvantage is that this kind of simulations consumes much more time and powerful computational resources are required to achieve it. The degree of development of these tools has been directly related to the progress in computer systems, so they still have some limitations in their applicability, mainly due to reasons of processing time. Despite the complexity of diesel combustion phenomenon, it has been the subject of numerous studies that have contributed to increase its understanding (Miles, P., et al 2001), (Baumgarten, C., 2006). CFD simulations of diesel combustion engines had experienced a significant advance. However, there are still numerous issues to resolve, primarily regarding the exact description of the turbulence structure of during combustion and the selection of a suitable mesh size (Merker, A et al., 2004). The development of such tools represents a step forward in the design and optimization of combustion engines, to the point of changing the way in which researchers conceive the phenomenology of this process. In the last decade, some models for describing turbulence have been developed as the Large Eddy Simulation (LES) method for reactive flows, which can be applied to turbulent flows in complex geometries and the Reynolds-Averaged Navier-Stokes (RANS) method, that can model disturbances (vortices) at large scale (Miles, P., et al 2001) , (Basha, et al., 2009) Furthermore, LES method not only allows more precise estimations, but offers the possibility to analyze the interactions between combustion and acoustics, leading to an interest of the automotive industry to solve problems related to combustion instability (Basha, et al., 2009) Nowadays, efforts are focused to reach a spatial resolution that allows the application of direct numerical simulation (DNS), which has been to fully calculate flames in very simple geometries (Westbrook, 2005) .

On the other hand, there is another point of view in simulating the diesel engine performance by means of predictive modeling. This approach is based on semi-empirical correlations for heat release rate (as the so well-known Watson's Law) to estimate the pressure and temperature profiles. Typically, in this kind of models (classified as zero-dimensional or quasi-dimensional) the combustion chamber is discretized into one or two zones. Different submodels, with certain grade of complexity, are incorporated to describe for example heat transfer, turbulence or the deviation from ideal gas. The indicated cycle of the engine is the graphical representation (usually the pressure vs. crank angle diagram) of the duty cycle at real operating conditions. From this diagram volume and other parameters are calculated to quantify the engine performance. As this quantification is based on in-cylinder parameters, actual parameters would be different due to friction in the piston rod - crank mechanism and heat and mass losses.

1.1 The predictive modeling

A predictive model called "SICICLO" has been used to determine the pressure and temperature profiles from the heat release curve. This model is based on the following assumptions:

- Pressure is assumed to be uniform within the combustion chamber.
- The evolving fluid within the cylinder is a mixture of air, fuel and combustion products stoichiometrically burnt.
- This mixture has ideal gas behavior.
- The internal energy of each species in the mixture is assessed at the average temperature inside the cylinder.
- The heat losses through the walls of the cylinder are calculated using the model proposed by Woschni (Heywood, 1998)

The input data for the model are: the geometric characteristics (e.g. piston diameter, stroke, connecting rod length, displacement, number of cylinders), operating conditions (e.g. rotational speed, equivalence ratio, initial temperatures and pressures), and some parameters associated with the iterative process. With this set of input data, the model is able to predict the temporal evolution of the heat release rate, and from this, the pressure profiles and the instantaneous average temperature inside the combustion chamber. Additional data of interest, concerning the indicated cycle, such as work and power, average pressure, fuel consumption, efficiency, peak pressure and peak temperature are also calculated.

The key equation of the model is the heat release fraction, which is based on an empirical relationship known as the Watson's law (1), which is composed of the heat released factions of the premixed (2) and the diffusion (3) phases.

$$FQL = \beta * FQL_p + (1-\beta) * FQL_d \tag{1}$$

$$FQL_p = 1 - \left(1 - \left(\frac{\alpha - \alpha_0}{\Delta\alpha_c}\right)^{C3}\right)^{C4} \tag{2}$$

$$FQL_d = 1 - \exp\left(-C1\left(\frac{\alpha - \alpha_0}{\Delta\alpha_c}\right)^{C2}\right) \tag{3}$$

where FQL is the fraction of heat released, β is the proportion of burnt fuel during premixed combustion, α is the crankshaft position (in degrees), α_0 is the angle of fuel injection, $\Delta\alpha_c$ the duration of the combustion, C_1 the degree of completion of the combustion (taken as 6.908), C_2 is a shape parameter (taken as 2.2), C_3 is the peak shift (taken as 2.2) and C_4 is a constant (recommended in 5000) (Heywood J.B., 1998) The subscripts *p* and *d* corresponds to the premixed and diffusion phases, respectively

1.2 CFD simulation of diesel combustion

By means of CFD simulation, the equations that govern the behavior of a flow, even for complicated geometries, are numerically solved. The result is reflected in the temporal and spatial history of different variables involved in the particular problem. To obtain the numerical solution of the set of partial differential equations that govern the flow, whatever its nature is, they are approximated into a set of algebraic equations. Most commercial CFD codes are based on the FV method discretization which is a general formulation of the finite differences method. The conservation equations are treated in differential form and resolved on discrete control volumes, using finite difference type approximations (Versteeg, et al., 1995)

The diesel combustion simulation in the present work was carried out using the CFD code OpenFoam, which is based on the Finite-Volume (FV) method to solve the transport equations governing the flow. Since the process involves two-phases, namely, the air sucked into the engine and the liquid fuel, the Eulerian – Lagrangian formulation has been chosen to address the problem (Westbrook, Ch., et al 2005)

The equations used in the CFD simulation are: Equation of Continuity (4), Equation of momentum (5), Energy equation (6) and Equations of turbulence (7) and (8) (Nordin, Niklas., 2001)

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{u}) = \nabla \cdot \left[\rho D \nabla\left(\frac{\rho_m}{\rho}\right)\right] + f_m + \rho^{s_m}\ \delta_{ml} \tag{4}$$

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\sigma} + \mathbf{F_s} + \rho\, g - \frac{2}{3}\nabla(\rho k) \tag{5}$$

where p_m is the species density, p the gas density, u the gas velocity, f_m is the source term for chemical effects, D is the diffusivity coefficient, p^{sm} is the term for liquid evaporation, p is the gas pressure, σ is the viscous stress tensor, F_s is the rate of gain/loss of momentum due to the spray per unit of volume, g is the body forces (considered as constant).

$$\frac{\partial(\rho e)}{\partial t} + \nabla \cdot (\rho \mathbf{u} e) = -p\nabla \cdot \mathbf{u} - \nabla \cdot \mathbf{J} + \rho\,\varepsilon + \dot{Q}^c + \dot{Q}^s \quad (6)$$

Where e is the specific internal energy, and J is the heat flux vector, which is the contribution of the heat by conduction and diffusion enthalpy.

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{u} k) = -\frac{2}{3}\rho k \nabla \cdot \mathbf{u} + \sigma : \nabla \mathbf{u} + \nabla \cdot \left[\left(\frac{\mu}{\mathrm{Pr}_k}\right)\nabla k\right] + \rho\varepsilon + W^s \quad (7)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{u} \varepsilon) = -\left(\frac{2}{3}C_{\varepsilon 1} - C_{\varepsilon 3}\right)\rho\varepsilon \nabla \cdot \mathbf{u} + \nabla \cdot \left[\left(\frac{\mu}{\mathrm{Pr}_\varepsilon}\right)\nabla \varepsilon\right] + \frac{\varepsilon}{k}C_{\varepsilon 1}\boldsymbol{\sigma} : \nabla \mathbf{u} - C_{\varepsilon 2}\rho\varepsilon + C_s W^s \quad (8)$$

Among the RANS models for turbulence, the most popular are the standard k-ε, the RNG k-ε, the realizable k-ε and Reynolds Stress Model (RSM). k and ε refer to the turbulent kinetic energy and its dissipation rate, respectively. In this work, the k-ε model has been chosen because of its robustness, simplicity and reasonable accuracy.

In the same way, different models for atomization, secondary breakup, evaporation, ignition and combustion have been incorporated. The complete formulation and validation is detailed in (Gutiérrez, E. I., 2008), (Stiesch, G., 2005)

2. SIMULATION CONDITIONS

A series of tests at different engine speeds and loads were conducted on a direct injection diesel engine mounted on a test bench (see Table 1) in order to characterize its mechanical and environmental behavior.

Table 1. *Diesel engine specifications*

Reference	ISUZU 4JA1
Type	Direct injection diesel
Feeding	Turbocharged
Displacement	2499 cm^3
Alignment	4 cylinders in line
Diameter / stroke	93 mm / 92 mm
Compression ratio	18.4:1
Maximum power	59 kW (80 hp) @ 4100 rpm
Maximum Torque	170 Nm @ 2300 rpm
Injection pump	Mechanical rotary

Pressure in the combustion chamber, flows of air and fuel, exhaust gas temperature, intake air temperature, lubricant temperature, shaft rotation speed, and emissions of O_2, CO, CO_2, HC, NOx and particulate material were measured. Four test conditions, at a constant engine speed of 2000 rpm (see Table 2), were chosen for the analysis and comparison of the CFD code OpenFoam and the predictive model SICICLO.

Table 2. *Operating conditions of the engine simulated*

Parameters	Simulations			
	S1	S2	S3	S4
Engine load (N.m)	20	40	60	80
Intake pressure (kPa)	103	110	112	118
Intake temperature (K)	321	325	328	334
Injected fuel mass (mg)	2.02	2.81	3.54	4.40
Start of injection (SOI) (Degrees)	-23°	-24°	-25°	-28°
Duration of injection (Degrees)	15°	15°	15°	15°

In the case of the CFD simulation, it was only considered the part of the cycle in which the system is thermodynamically closed, i.e, between the closing of the intake valve and the opening of the exhaust valve.

3. EXPERIMENTAL CONDITIONS

The experimental design was a factorial type (2k), with engine loads of 20 Nm, 40 Nm, 60 Nm and 80 Nm as factors and diesel fuel and biodiesel fuel as factor levels, being the response variable the engine mechanical performance, in terms of its thermal efficiency. In this paper only the diesel fuel data was analyzed.

The experimental data was obtained from a diesel engine (ISUZU AJ1) test bench equipped with a Eddy current dynamometer with an output range up to 120 kW (see Figure 1). The test bench allows measuring the main engine development parameters in real-time such as: torque-speed, power-speed, specific fuel consumption (by gravimetry +/- 0.1 g), air flow (Thermatel TA2 +/- 2 m/s), and thermal efficiency. Pollutant emissions (with an AVL Dicom 4000; CO +/- 0.01% vol, NOx +/- 10 ppm, HC +/- 1 ppm) and opacity (AVL Dismoke 4000 +/- 1%) also were measured. The pressure inside of the combustion chamber (CCP), which is the most important parameter of the present study, was measured using a piezoelectric sensor (Kistler 5011B with sensitivity = 20 pc/bar), installed into one of the cylinders and connected to signal amplification and conditioning equipment.

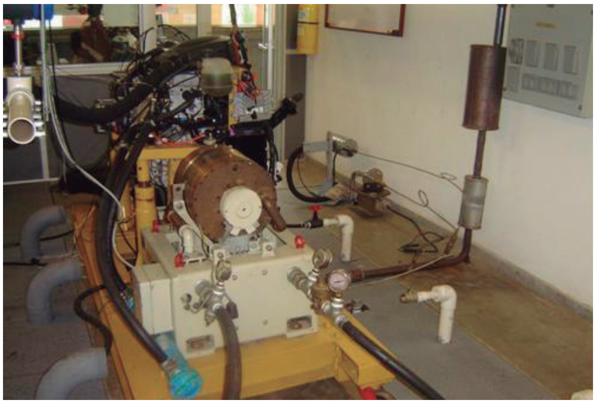

Figure 1. *Diesel engine test bench*

During the experiments, the steady state operation of the engine was guaranteed by the exhaust gases temperature stability, as proposed in (Armas, O., et al., 2013), (Yehliu, K., et al., 2013)

4. RESULTS

The evolution of the pressure inside the combustion chamber is the main input for the thermodynamic analysis of the duty cycle of an internal combustion engine, because its variation determines how much work is done. The experimental and simulated pressure profiles are compared in Figures 2 to 5 for the selected operating modes. There is a very good agreement between simulations and experiments, especially at high loads. Although a slight tendency to overestimate the pressure can be observed, the peaks of the curves are similar for the four evaluated modes. The differences obtained may be attributed to the fact that in the simulation the blow-by mass, and exhaust gas recirculation (EGR) were not considered. Additionally, there could be some misalignments in the heat transfer model in the case of SICICLO simulation.

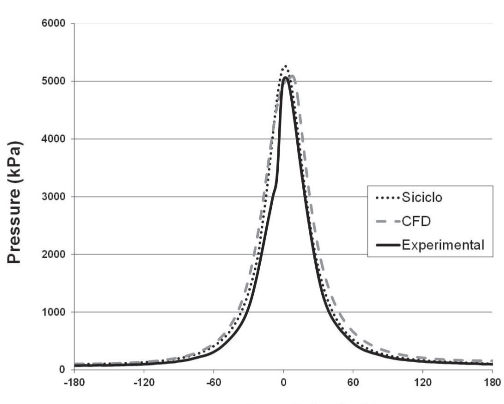

Figure 2. *Pressure profile at 2000 rpm and 20 Nm load*

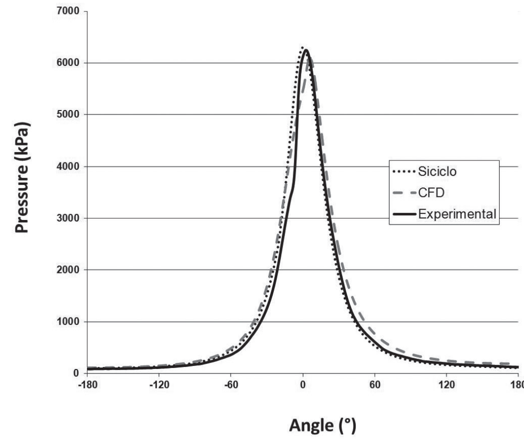

Figure 3. *Pressure profile at 2000 rpm and 40 Nm load*

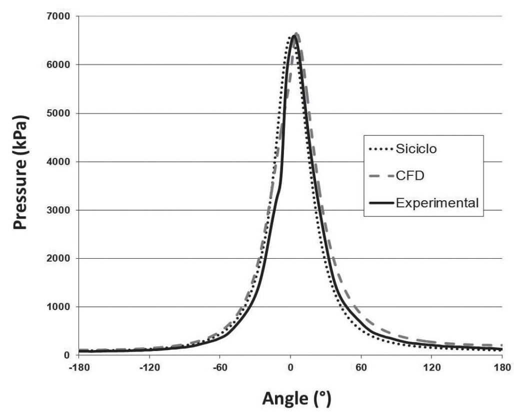

Figure 4. *Pressure profile at 2000 rpm and 60 Nm load*

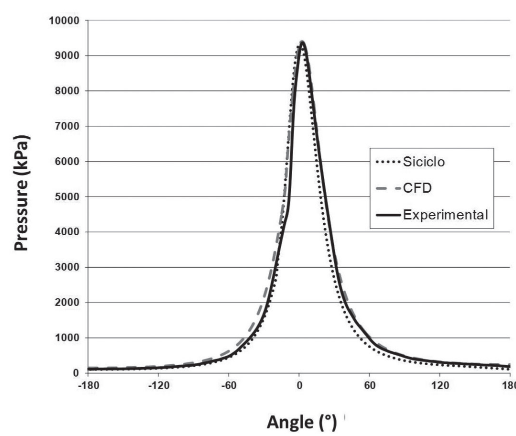

Figure 5. *Pressure profile at 2000 rpm and 80 Nm load*

Temperature inside the combustion chamber provides a first idea of how the combustion is taking place within the engine. Peak values may eventually influence the formation of contaminant species such as nitrogen oxides (NO_x). In Figures 6 to 9 it is shown the comparison between the temperature profiles obtained with CFD and those obtained by means of SICICLO (i.e. from the measured pressure curve and the thermodynamic model of one zone) for the four operating modes. It is observed that, except for the high-load case (80 Nm), there is a clear trend by the CFD simulation to overestimate the temperature during the compression stroke, underestimate it once combustion begins and overestimate it during expansion. This phenomenon may be due to several reasons such as non-uniform surface temperature of the inner walls of the combustion chamber as assumed in the simulations as well as possible mismatches in the combustion model and the injection characteristics (e.g. pressure, velocity, start and duration of injection). However, the temperature peaks were comparable in magnitude and location for the four evaluated cases.

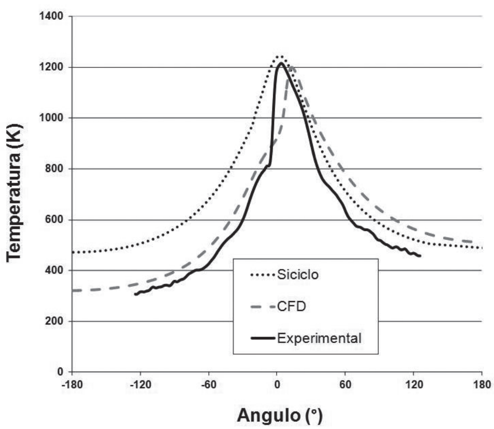

Figure 6. *Temperature profile at 2000 rpm and 20 Nm load*

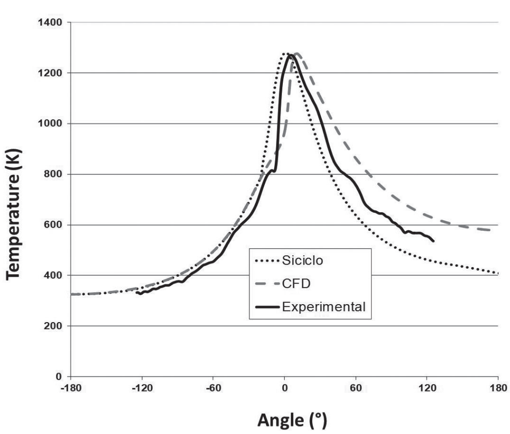

Figure 7. *Temperature profile at 2000 rpm and 40 Nm load*

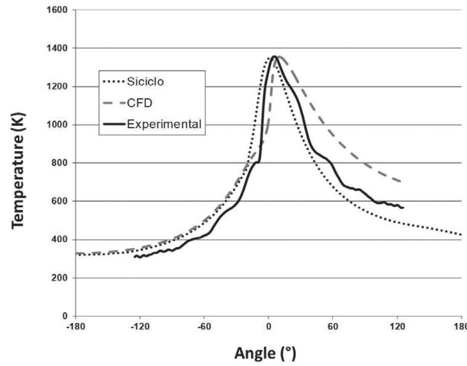

Figure 8. *Temperature profile at 2000 rpm and 60 Nm load*

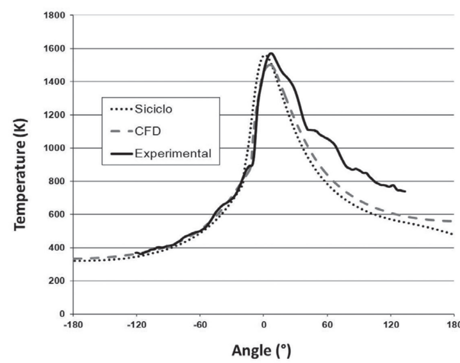

Figure 9. *Temperature profile at 2000 rpm and 80 Nm load*

The fraction of heat release (FQL) is the integral of the heat release rate normalized to the product of the mass of fuel injected and the calorific value of the fuel (in mass basis). FQL qualitatively allows determining the duration of the combustion and the fuel burn rate. In Figures 10 to 13 show the FQL results for the four evaluated cases. In all cases, it is observed a difference proportional to the change in temperature. However, the profiles have the same shape and tend to converge to the same value. The difference in the trends leads to a difference in the measurement of the duration of combustion.

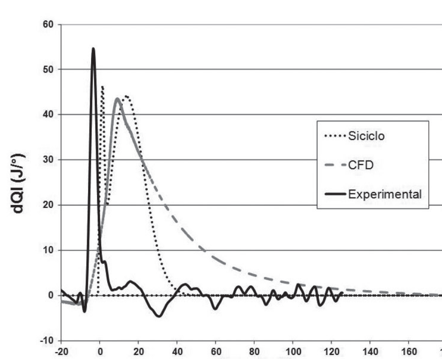

Figure 10. *Heat release rate profile at 2000 rpm and 20 Nm load*

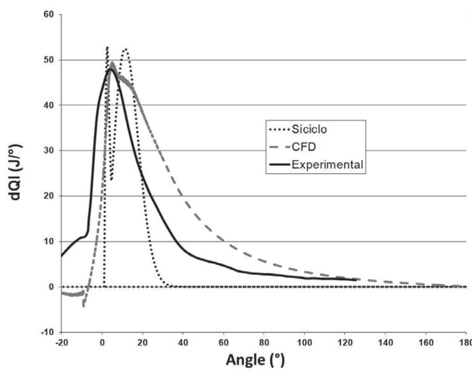

Figure 11. *Heat release rate profile at 2000 rpm and 40 Nm load*

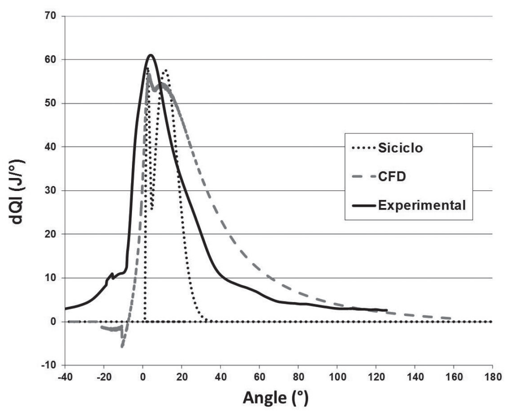

Figure 12. *Heat release rate profile at 2000 rpm and 60 Nm load*

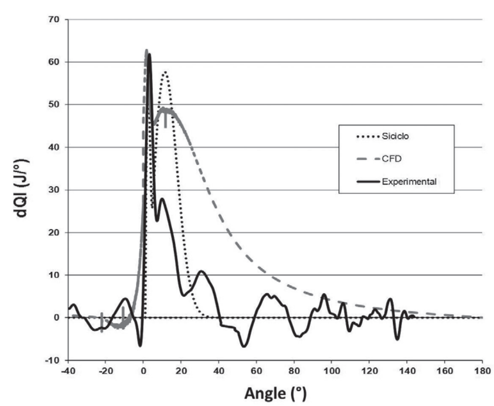

Figure 13. *Heat release rate profile at 2000 rpm and 80 Nm load*

The indicated parameters are those that show the engine thermodynamic performance, but seen from the inside of the combustion chamber, so that, in any case always they will be more optimistic than those measured as actual parameters. Usually, the parameters reported are: indicated work, indicated power, indicated efficiency, indicated fuel consumption (IFC) and indicated mean pressure (IMP). The indicated work is obtained by numerical integration of the measured pressure profile in the combustion chamber. Table 3 shows the results obtained by the numerical simulation with SICICLO and the measurements of pressure and angle data.

Table 3. *Indicated parameters for the cycle*

2000 rpm and 20 Nm load		
	CFD Simulation	**Experimental**
Indicated work (kJ/cycle)	0.31	0.26
Indicated Power (kW)	4.9	4.4
IFC (g/kW-H)	476	491
Indicated Efficiency (%)	14.3	13.5
IMP (MPa)	118	106
2000 rpm and 40 Nm load		
	CFD Simulation	**Experimental**
Indicated work (kJ/cycle)	0.61	0.53
Indicated Power (kW)	9.7	9.1
IFC (g/kW-H)	384	395
Indicated Efficiency (%)	19.8	19.1
IMP (MPa)	233	219
2000 rpm and 60 Nm load		
	CFD Simulation	**Experimental**
Indicated work (kJ/cycle)	0.91	0.80
Indicated Power (kW)	14.2	13.3
IFC (g/kWh)	280	312
Indicated Efficiency (%)	29	26.4
IMP (MPa)	341	320
2000 rpm and 80 Nm load		
	CFD Simulation	**Experimental**
Indicated work (kJ/cycle)	1.23	1.10
Indicated Power (kW)	19.5	18.3
IFC (g/kW-H)	206	229
Indicated Efficiency (%)	39.5	36.3
IMP (MPa)	468	439

It is observed that in all cases the listed parameters were overestimated by the simulation when compared with the measured data. This can be explained by the fact that the simulations only considered the corresponding cycle of compression and expansion strokes, while the actual cycle was complete, including intake and exhaust strokes, which thermodynamically represent negative work (pumping work), and increases as the level of motor load does.

5. CONCLUSIONS

For the four evaluated cases, simulated average pressure profiles inside the combustion chamber matched reasonably well to those measured on a test bench, particularly at high engine load.

By contrast, simulated temperature profiles presented some differences with those obtained from the experimental pressure profile when comparing the shapes. However, maximum temperature peaks and their localization were comparable for all the considered cases. The differences may be due to mismatches in the model for heat transfer losses to walls or mismatches in the thermodynamic model obtained with the experimental temperature.

Curves of fraction of heat released (FQL) also showed differences, due to strong dependence with temperature. However it seems that the CFD simulation predicts much better the performance parameters, whose shape is similar to that predicted by theory, in regard to the peaks of premixed and diffusion heat release. This determines differences in the calculation of parameters related to combustion as the delay time, the start of combustion and combustion duration, among others.

Taking in account that indicated parameters of engine are based on the evolution of the pressure inside the combustion chamber, the results obtained by the simulation are very similar to those obtained from the measured pressure in the combustion chamber.

It is demonstrated the convenience and usefulness of computational tools for modeling processes into the cylinder of an internal combustion engine, even in the case of diesel engines the turbulence modeling continues to represent a serious challenge and it is an issue that must improve significantly.

6. REFERENCES

MERKER, G., SCHWARZ, C., STIESCH, G., OTTO, F., (2004); Simulating Combustion. Simulation of combustion and pollutant formation for engine-development. Editorial Springer. Berlín, 72-86.

MILES, P., MEGERLE, M., SICK, V., Richards, K. et al., (2001); The Evolution of Flow Structures and Turbulence in a Fired HSDI Diesel Engine, SAE Technical Paper,1-9.

BAUMGARTEN, C., (2006); Mixture Formation in Internal Combustion Engines, Editorial Springe., Berlin, 32-38.

BASHA, S., Raja GOPALA, K., (2009); In-cylinder fluid flow, turbulence and spray models—A review. Renewable and Sustainable Energy Reviews, Volume 13, Issues 6-7, 1620-1627.

WESTBROOK, Ch., MIZOBUCHIBO, Y., et al (2005); Computational Combustion, Proceedings of the Combustion Institute, Volume 30, Issue 1, 125-157.

Heywood J.B., (1998); Internal Combustion Engine Fundamentals, Editorial McGraw-Hill, New York, 676-682.

VERSTEEG H., MALALASEKERA W., (1995); An Introduction to Computational Fluid Dynamics. The Finite Volume Method, 1a edición, Longman Scientific & Technical, London, 2-7.

NORDIN, Niklas., (2001); Complex Chemistry Model of Diesel Spray Combustion PhD thesis, Chalmers University of Technology, Goteborg, 4-15.

GUTIÉRREZ, E. I., (2008); Modelado Numérico de la Atomización y Evaporación de un Chorro Diesel Mediante Análisis CFD, Tesis de Maestría, Universidad de Antioquia, Medellín, Colombia, 13-19.

STIESCH, G., (2005); Modeling Engine Spray and Combustion Processes. Editorial Springer. Berlín, 109-112.

ARMAS, O., GARCÍA-CONTRERAS, R., RAMOS, A., (2013); Impact of alternative fuels on performance and pollutant emissions of a light duty engine tested under the new European driving cycle. Applied Energy, Vol. 107, Elsevier, England, 183-190.

YEHLIU, K., ARMAS, O., RANDY, L., VANDER Wal, R., BOEHMAN, A.L., (2013); Impact of engine operating modes and combustion phasing on the reactivity of diesel soot. Combustion and Flame, Vol. 160 (3), Elsevier, Pages 682-691.

7. ACKNOWLEDGEMENTS

The authors wish to thank the group Efficient Energy Management (GIMEL) of the Universidad de Antioquia, where experiments were conducted in the engine test bench.