EXPERIMENTAL AND NUMERICAL ANALYSIS OF SPRAYS FROM A CR DIESEL INJECTOR

Allocca L. *, De Vita A. **, Rosa P. **

***Istituto Motori CNR, Via Marconi, 8, 80125 Napoli, ITALY**
**** Dip. Energetica, Università de L'Aquila Loc. Monteluco, 67040 Roio Poggio (AQ), ITALY**

Abstract

Experimental and numerical study of diesel sprays from a Common Rail (CR) injection system has been performed. A suitable interaction between numerical and experimental analysis has allowed to develop and to validate a model for the initial break-up process of the liquid fuel jet.
Such process is very important, more than the secondary break-up, to better understand the complex physical phenomena related to high pressure sprays impacting on combustion chamber walls of small diesel engines.
Sprays, from a single hole high pressure electronic control injector, have been visualized and analysed using a light sheet technique based on a Nd-YAG pulsed laser, a synchronized CCD camera and a digital image processing software.
The developed numerical tool is a turbulent induced atomisation model based on the K-H instabilities theory, and implemented in the KIVA3 CFD code.
Shape, structure, temporal and spatial evolution of the spray have been analysed varying the injection pressure and the backpressure of the gas in an optically accessible test chamber.
The predictive capability of the model has been evaluated by comparison with experimental data at the different operating conditions.

NOMENCLATURE

D	droplet diameter
D_{32}	Sauter mean diameter
k	kinetic energy
L	length scale
p	pressure
r	droplet radius
t	time
U	droplet-gas relative velocity

Greek letter

ε	rate of kinetic energy dissipation
ρ	density
σ	surface tension
τ	time scale

Subscripts

A	atomisation
a	after break-up
avg	average
b	before break-up
exp	exponential growth
ch	*test* chamber

g	gas
inj	injection
l	liquid
spn	spontaneous growth
t	turbulent
w	wave

1. Introduction

Modern high pressure electronically controlled Common Rail injection systems are successfully used in diesel engines, allowing to shape the injection curve, to reduce emissions, to get a better fuel economy and to provide more comfort. Due to the high injection pressure, the velocities of the droplets emerging from the nozzle assume values greater than 300 m/s and strongly atomise. In small size engines, a large amount of fuel reaches the wall of the combustion chamber, so the spray structure and the wall-impingement assume a relevant role in the short time available to prepare the mixture ready to be burned before the start of ignition[1-5].

Jet impingement on a flat wall expands the total volume occupied by the fuel, producing a lot of new droplets and increasing the gas entrainment in the spray [6-8]. It behaves in a significantly different way depending on the injection pressure, the backpressure in the chamber and the wall temperature [5-7, 9].

In the last years a lot of numerical models have been developed to simulate fuel injection systems and spray structures. They are very important tools to better understand the fuel injection process and to study different engine geometries, operating conditions and injection strategies, reducing the experimental activity. [10-17]

The recent development of higher pressure injection systems requires further efforts in modelling activities to simulate very complex physical phenomena: disintegration process from liquid fuel to a spray, structure, droplet size and velocity of the spray impinging on the piston bowl wall, spray evolution after impingement, etc.

An interaction between numeric computation and experimental analysis has been performed to develop an atomisation model able to simulate the spray formation and penetration, and to predict the droplet size and velocity of the impinging spray.

The experimental apparatus and the model are described and the experimental and numerical data are compared and discussed.

Results provide useful information for a better understanding of the phenomena related to the high-pressure sprays produced by common rail injection systems.

2. Numerical code

The description of a liquid jet is one of the greatest difficulties in CFD simulations. A typical Diesel nozzle injects a coherent liquid column into the chamber that, because of its internal turbulence and cavitation and because of the interaction with the gaseous phase, breaks up into drops and ligaments varying greatly in size.

A break-up model has been implemented into the CFD KIVA code, which solves the three-dimensional equations of chemically reactive flows with sprays.

The fuel jet is modelled, by the KIVA code, as a train of parcels of drops (blobs) with a Sauter Mean Diameter equal to the nozzle diameter. Each computational parcel represents a group of physically similar droplets.

In the present work a Rosin-Rammler distribution is proposed to take into account the distribution of the parcels outgoing the nozzle. As know, its cumulative expression is:

$$V = 1 - \exp\left(-\frac{D^q}{\overline{\overline{D}}} \right) \qquad (1)$$

where q is the distribution parameter, that in the present case is equal to 5, and

$$\overline{D} = D_{32}\Gamma\left(1 - q^{-1}\right) \tag{2}$$

where G is the gamma function and D32 is the Sauter Mean Diameter.
Primary blobs may undergo primary break up [18].
Depending on its main reasons, primary break up can be classified in:

- Aerodynamic-induced break-up. The main phenomenon is the Kelvin-Helmotz instability, which leads to the formation of waves on the liquid jet surface, caused by the relative motion between the gas and the injected fuel. In terms of dimensionless parameters, the Weber number determines the growth rate of these waves and finally the disintegration of the jet into smaller droplets.
- Turbulence-induced break-up. For fully turbulent flow conditions in the injection holes, the radial velocity component soon leads to the disruption of the surface film, followed by general disintegration of the jet. Even when the jet is injected into vacuum, it will disintegrate solely under the influence of its own turbulence. According to this model, the turbulent fluctuations within the liquid jet emerging from the injection hole are the producers of the initial surface perturbations, which are assumed to occur predominantly at a certain characteristic wavelength proportional to the integral length scale of turbulence.
- Cavitation-induced break-up. When cavitation is initiated in the injection holes, dramatic changes occur in the spray structure. Since the pressure around the emerging jet is much higher than the pressure inside the cavitating bubbles, these gradually collapse. This process causes perturbations, formed on the surface of the liquid jet, which at the time of total collapse or at the time at which the bubbles reach the jet surface, leads to the jet disintegration and formation of smaller droplets.

All the previous phenomena are simultaneously present, even if with different intensities, and they determine the spray characteristics together. In the present work, cavitation effects can be neglected because in the single-hole injector the fuel in the nozzle goes through a straight line and there is no bends that can produce local pressure drop. For these reasons, only the *turbulence induced atomisation model* has been implemented.
In turbulence induced atomisation model the jet internal turbulence and gas inertia are thought to be the two dominant break-up forces. The turbulent fluctuations in the emerging jet are responsible of the initial surface perturbations that grow exponentially by K-H instabilities due to the interaction with surrounding gas. The choice of these two forces is the result of an order magnitude analysis of the possible relevant forces in the atomisation regime and experimental observation that fluctuating eddy motion in the jet induce initial surface perturbations.
Break-up model is based on the calculation of the time scale, $t_{A,t}$, and length scale, $L_{A,t}$, of atomisation.
The time scale of atomisation is assumed to be the sum of two terms:

$$t_{A,t} = t_{spn,t} + t_{exp,t} \tag{3}$$

The length scale of atomisation is supposed to be proportional to the integral length scale of turbulence, L_t:

$$L_{A,t} = C_1 L_t \tag{4}$$

$t_{spn,t}$ is the time required for the spontaneous growth of instabilities while $t_{exp,t}$ is the time for their exponential growth until they detach as droplets. The first is proportional to the turbulent time scale, t_t, the latter to the wave growth time scale, $t_{w,t}$, provided by K-H instability theory:

$$t_{spn,} = C_2\, t_t \qquad t_{exp,t} = C_3\, t_{w,t} \tag{5}$$

$$t_{w,t} = \frac{1}{\sqrt{\dfrac{r_l r_g}{\left(r_l + r_g\right)^2}\left(\dfrac{U}{L_{w,t}}\right)^2 - \dfrac{s}{\left(r_l + r_g\right)\left(L_{w,t}^3\right)}}} \tag{6}$$

$L_{w,t}$ is the wavelength of surface instability, proportional to the integral length scale of turbulence:

$$L_{w,t} = C_4 L_t \tag{7}$$

The turbulent length and time scale are related to the average turbulent kinetic energy, k_{avg}, and its dissipation rate, ε_{avg}, in the injection holes:

$$L_t = C_m \frac{k_{avg}^{3/2}}{e_{avg}} \qquad t_t = C_w \frac{k_{avg}}{e_{avg}} \tag{8}$$

Break-up condition is:

$$t(i) \geq t_{A,t} \tag{9}$$

where $t(i)$ represents the time after the injection of the i^{th} computational parcel.
After break-up occurs, smaller droplets are formed whose characteristic radius is:

$$r_a(i) = r_b(i) - L_{A,t} \cdot \frac{t(i)}{t_{A,t}} \tag{10}$$

where subscripts *a* and *b* mean after and before break-up respectively.
The constants of the model are set to the values summarized in Tab.1.

C_1	4
C_2	1
C_3	3
C_4	8
C_m	0.09
C_v	70

Tab.1. Constant values used in the model

3. Experimental apparatus

The experimental apparatus, shown in fig. 1, consists of a constant volume spray chamber, a high pressure fuel injection system, an imaging system and a data acquisition and control system.
The spray chamber is an optically accessible vessel, controlled in pressure and temperature, equipped with three large quartz windows (80 mm diameter) and filled by inert gas (N_2). It has a volume of $5*10^{-3}$ m^3 and is capable of operating at pressure up to 5.0 MPa.
The sprays have been generated, in single-shot mode, by an electronically controlled Common Rail (CR) injection system. Open software, via an Electronic Control Unit (ECU), has allowed setting the injection pressure, the injection duration and the timing. An injection duration of 1.0 ms and injection pressures of 80 and 120 MPa have been set with delivered fuel quantities of 8.44 10^{-3} g/shot and 9.67 10^{-3} g/shot, respectively. The ISO 4113 calibration fluid has been used as standard diesel fuel. An axial single-hole mini-sac nozzle injector type has been used with the hole diameter and length of 0.18 and 1.0 mm, respectively.
The sprays, emerging co-axially with respect to the chamber, have been lightened, at different time from the start of injection (SOI), by a pulsed laser sheet, 80 μm thickness and 12 ns duration, generated by a Nd-YAG laser operating on its second harmonic. The images have been captured by a PULNIX TMC-6 CCD camera, synchronized with the laser pulse, providing images size of 768 x 568 pixels with 8 bits intensity resolution.
Spray structure and morphology, jet cone angles and spray penetrations have been obtained by suitable software developed to process the acquired images.
Further details about experimental apparatus and image processing procedures have been reported in [19-21].

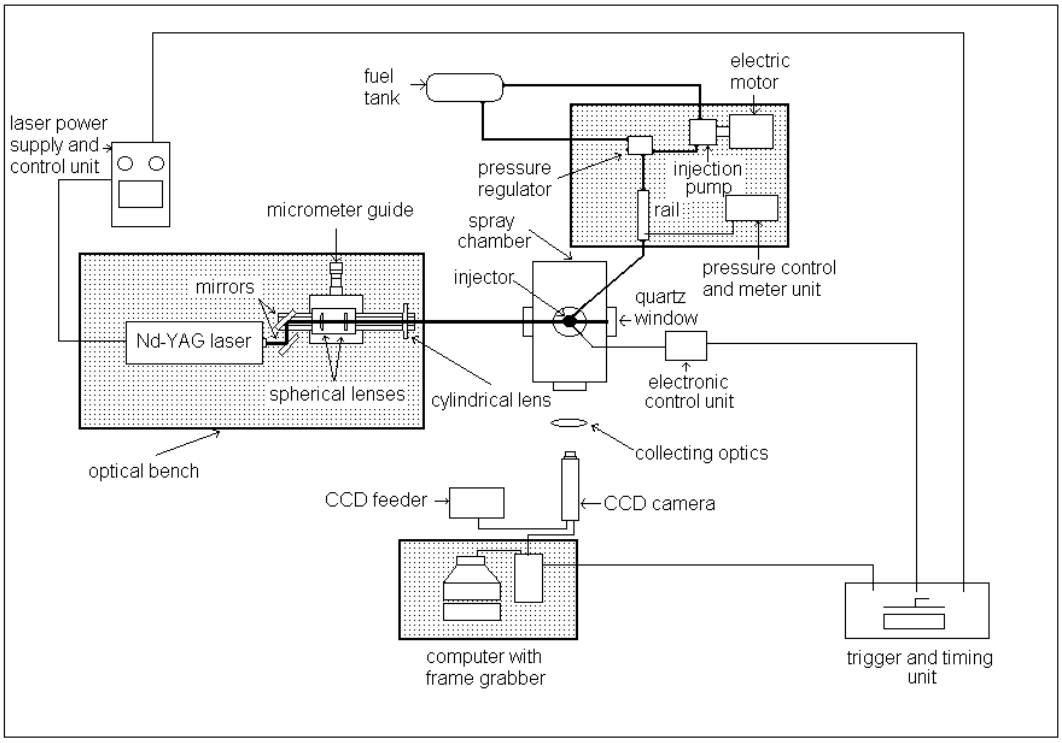

Fig. 1. Experimental apparatus

4. Results and discussions

1.1. Morphology and leading edge penetration data of spray

Fig. 2 shows images of spray evolution at different times after start of injection (SOI), being 120 MPa the injection pressure, 1.0 ms the injection duration and 0.1 MPa, 3.0 MPa and 5.0 MPa the backpressures. Sprays are characterized by high values of momentum and strongly penetrate the ambient gas, especially at backpressure of 0.1 MPa with constant spray velocities of about 300 m/s up to about 50 mm from the injector tip.

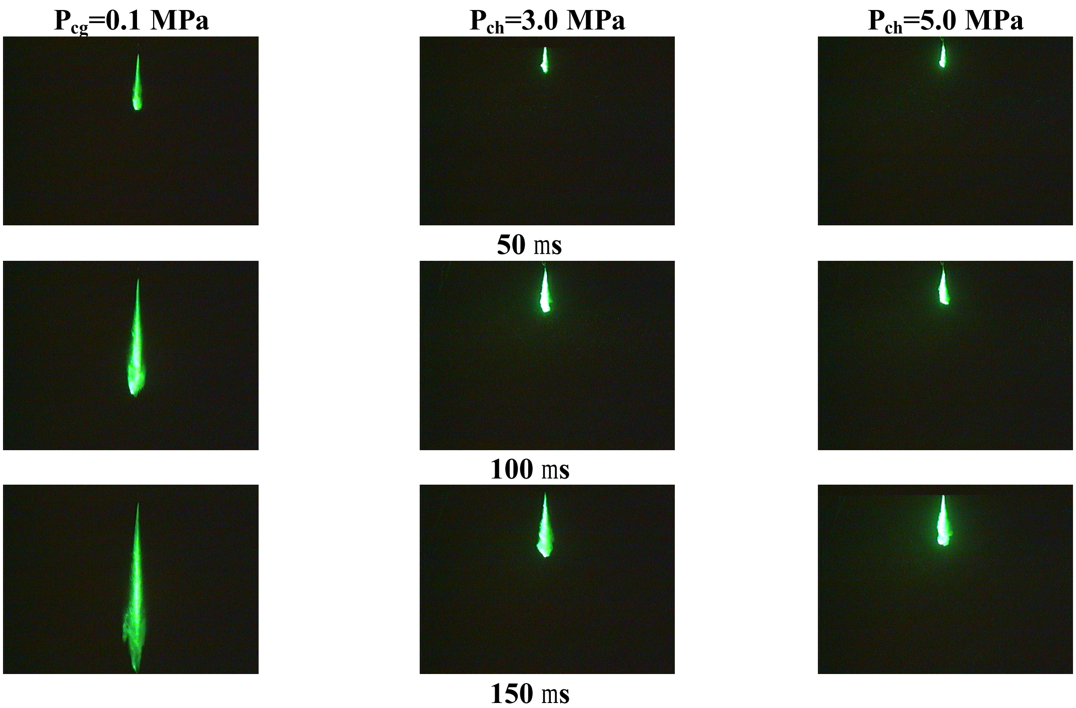

Fig. 2. Spray images at different backpressures and times after SOI. P_{inj}=120 MPa, t_{inj}=1.0 ms

The pseudo-colour conversion and binary transformation techniques [11] have been applied to the collected images to extract the shape and the section structure across the axial plane of the spray. The techniques allow to determine characteristic parameters of the phenomenon such as spray cone angle, spray leading edge penetrations and velocities.

In fig. 3 the conversion in pseudo-colour images of fully developed sprays are reported, showing structure and shape of evolving jet. The operating conditions are: 120 MPa injection pressure, 1.0 ms injection duration, 100 μs after SOI and 0.1 MPa, 3.0 MPa and 5.0 MPa backpressures.

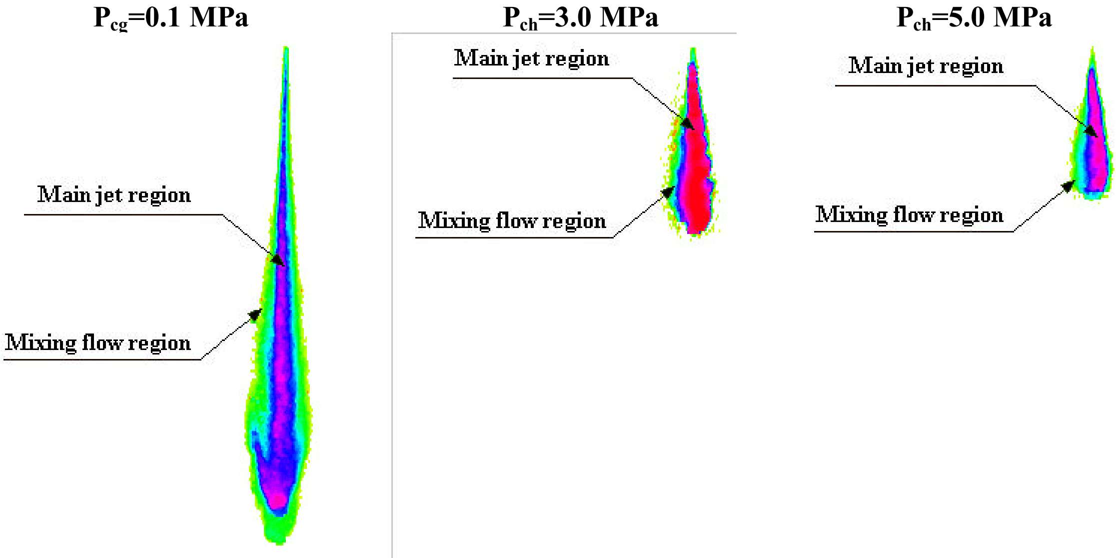

Fig. 3. Pseudo-colour images of the spray. P_{inj}=120 MPa, t_{inj}=1.0 ms. 100ms after SOI.

The spray consists of a "main jet region" and a "mixing flow region". The first region lies in the inside of the spray, it is characterized by high values of the fuel density and it is surrounded by finely atomised droplets, stripped in interaction with the gas in the chamber and forming the "mixing flow region". Ripples appear on the spray boundary due to its growth and interaction with the surrounding air flow characterized by reverse direction.

The solid cone structure of the spray, with high fuel density and high momentum, is evident. It is strongly affected by the pressure of the gas in the test chamber. Increasing the backpressure, spray penetration and velocity reduce, spray cone angle and radial distribution increase, the "mixing flow region" and ripples reduce.

Comparisons between spray evolutions at 120 MPa and 80 MPa injection pressures are shown in fig. 4.

Increasing injection pressure spray penetration slightly increases, being global spray structure and time evolution quite similar.

Fig. 5 reports, versus the time, spray leading edge penetration curves at the three different backpressures 0.1 MPa, 3 MPa, 5 MPa and at the fuel injection pressures of 80 MPa and 120 MPa.

The pressure in the test chamber strongly affects the spray penetration. For example at 100 μs after SOI and 120 MPa injection pressure, the changes of backpressure from 0.1 MPa to 3 MPa and to 5 MPa produce marked variations in the spray penetration, from 38.14 mm to 17.91 mm and to 14.51 mm, respectively.

At backpressure of 0.1 MPa the dominant mechanisms affecting the spray penetration are the nozzle opening and the very high momentum of the emerging droplets, while jet break-up phenomena don't seem to occur in the investigate range of the axial distance from the injector tip (0 - 50 mm) for any value of the injection pressure.

On the contrary, when backpressure increases at the values of 3,0 MPa and 5,0 MPa, blobs break-up clearly occurs at times after SOI less than 200 μs and markedly affects spray penetration and evolution.

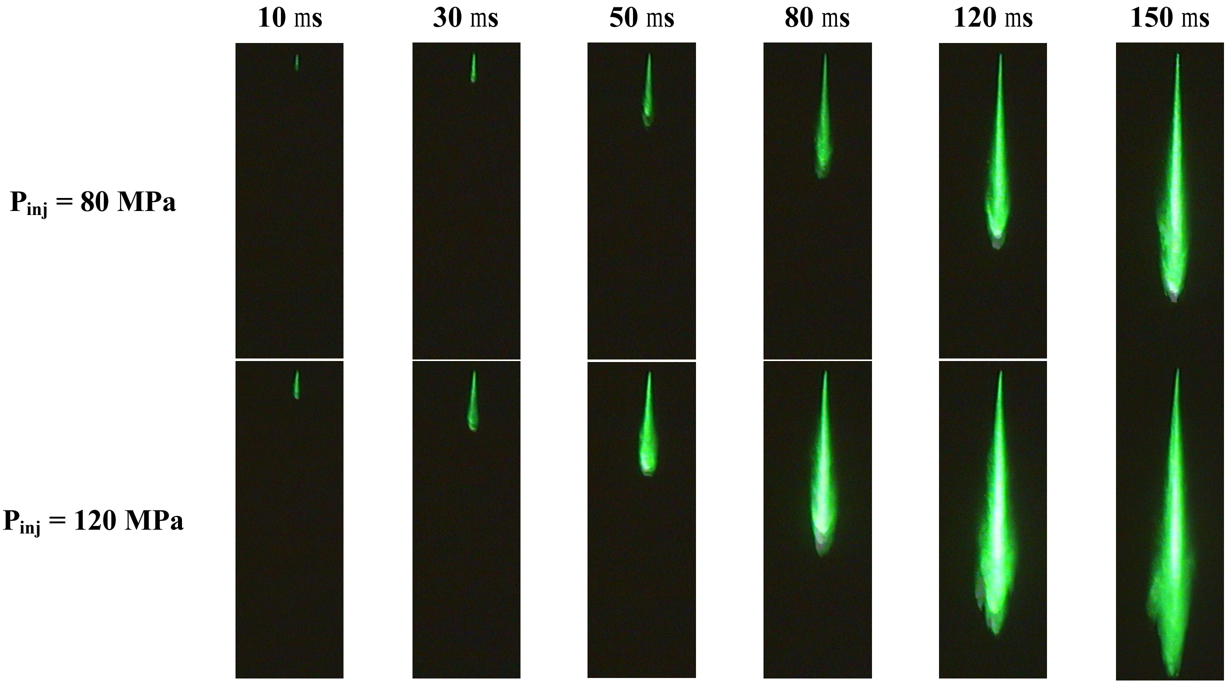

Fig. 4. Spray images at different times after SOI and 0.1 MPa backpressure

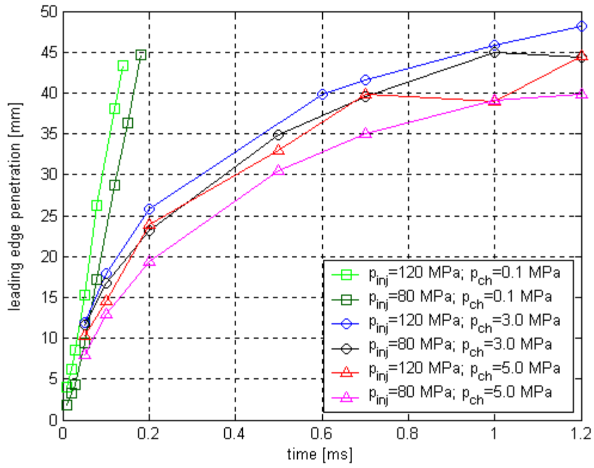

Fig. 5. Spray penetration curves at different backpressures and injection pressures

4.2 Numerical Results

The aim of the numerical work is to select and to validate proper models for the early phases of spray development and to set-up a model, implemented into KIVA 3V code, able to simulate spray structure and evolution, at different operating conditions. An interaction between numeric computation and experimental analysis has been performed to develop a model able to simulate the spray formation, structure and penetration, and to predict the droplet size distribution and velocity of the spray impinging on the piston bowl wall.

The numerical procedure is discussed in the paragraph "numerical model" and the parameters of the model are set at the values of tab. 1. Such values don't change varying the operating conditions: back- and injection pressure.

Computed spray evolutions and their comparison with experimental ones are reported in fig. 6, showing experimental and numerical spray images at same delay after SOI, being 80 MPa the injection pressure and 3 MPa and 5 MPa the backpressures.

Numerical and experimental spray penetrations are compared in the figures 7 and 8.

Fig. 7 shows experimental and computed penetration curves, being 80 MPa the injection pressure, 3 MPa and 5 MPa the ambient pressures.

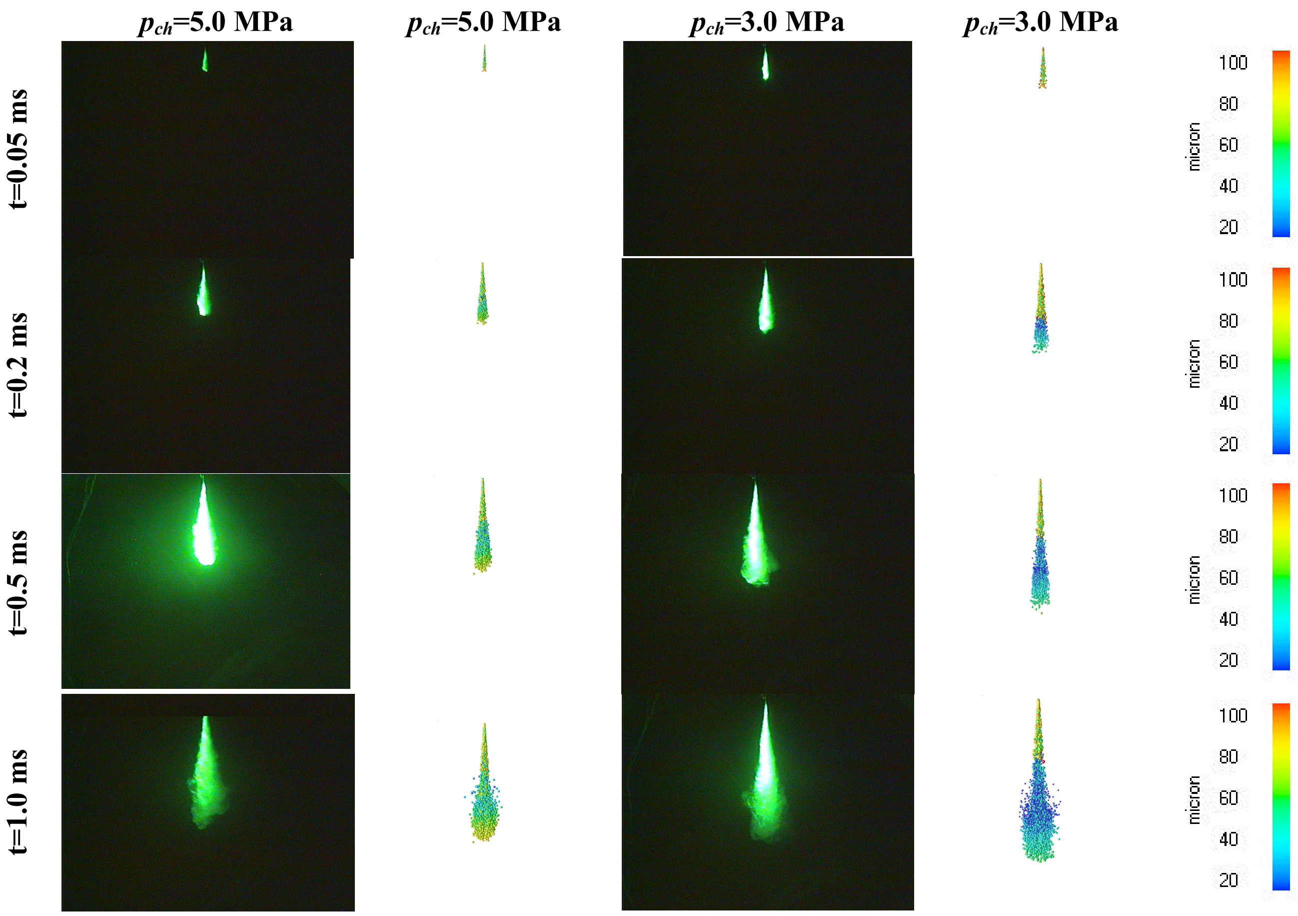

Fig. 6. Experimental and numerical spray images for different backpressures and 80 MPa injection pressure

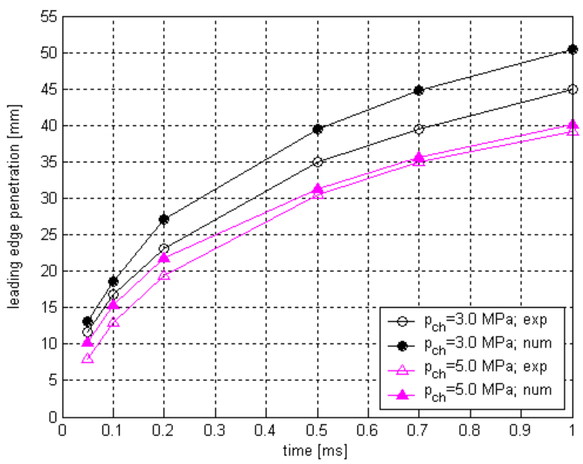

Fig. 7. Experimental and numerical penetration curves for different backpressures and 80 MPa injection pressure

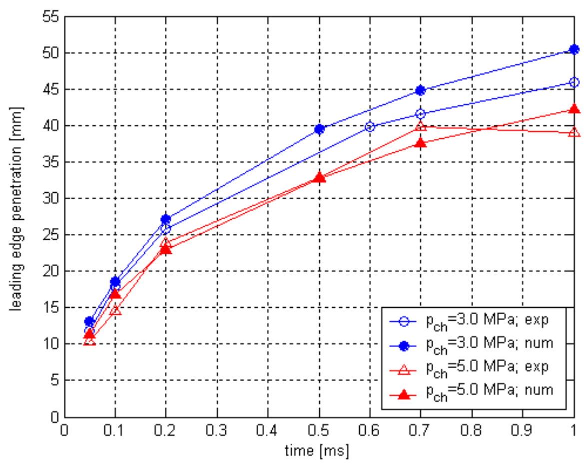

Fig. 8. Experimental and numerical penetration curves for different backpressures and 120 MPa injection pressure

In fig. 8 the experimental and numerical penetrations at 120 MPa injection pressure are compared.
The agreement between measurement and computation is good.
Some important physical phenomena are captured in the computations: the shape and the structure of the spray, the break-up process location, the spray variations due to the different values of the injection pressure and backpressure, the fuel-gas fluid-dynamic interaction with the reduction of droplets momentum and the decreasing of the penetration, the radial (normal to the spray axis) spread of the spray due to the increased pressure into the vessel.

So the proposed numerical procedure, able to capture some physical phenomena, experimentally available, allows to predict the droplet size distribution and velocity in spray sections at different axial distances from the injector tip. It is very important in order to study, to simulate and to better understand the complex phenomena related to the wall impingement and post- impingement evolution of the spray.

5. Conclusions

Experimental and numerical study of sprays from a Common Rail (CR) injection system for diesel engines has been performed. The aim was to develop and to validate a model for the initial break-up process of the liquid fuel jet. Such process is very important to better understand the complex physical phenomena related to high pressure sprays impacting on combustion chamber walls of small diesel engines.

A non-evaporative fuel was injected into a high-pressure test chamber, at room temperature and quiescent gaseous environment, from a single hole high pressure electronic control injector. Sprays has been lightened by a pulsed laser sheet from a Nd-YAG at 532 nm (80 μm thickness and 12 μs duration) and acquired by a CCD camera at different time from the SOI.

Using processing techniques of the collected spray images, shape, structure, temporal and spatial evolution of the spray have been analysed varying the injection pressure and the backpressure of the gas in the optically accessible test chamber.

The developed numerical tool is a turbulent induced atomisation model based on the K-H instabilities theory, and implemented in the KIVA3 CFD code.

The predictive capability of the model has been evaluated by comparison with experimental data at the different operating conditions.

The results may be summarized as follows:

1. the experimental set up and the used image processing technique have enabled to obtained detailed information on the shape, structure and morphology of the spray, allowing, also, a quantitative description, in terms of spray cone angle, leading edge penetration and velocity.
2. The spray consists of a “main jet region” and a “mixing flow region”. The first region lies in the inside of the spray, it is characterized by high values of the fuel density and it is surrounded by finely atomised droplets forming the “mixing flow region”.
3. The solid cone structure of the spray, with high fuel density and high momentum, is strongly affected by the pressure of the gas in the test chamber. Increasing the backpressure, spray penetration and velocity reduce, spray cone angle and radial distribution increase, and the “mixing flow region” reduces.
4. The effects of injection pressure are less evident. Increasing injection pressure spray penetration slightly increases, being global spray structure and time evolution quite similar.
5. The agreement between experimental and numerical results is good. Some important physical phenomena are captured in the computations: shape and the structure of the spray, break-up process location, spray modification due to different values of injection pressure and backpressure, fuel-gas fluid-dynamic interaction, radial (normal to the spray axis) spread of the spray due to the increased pressure into the vessel.

The proposed numerical procedure allows to predict the droplet size distribution and velocity in spray sections at different axial distances from the injector tip, important to study, to simulate and to better understand the complex phenomena related to the wall impingement and post- impingement evolution of the spray.

References

[1] Boehner W, Hummel K (1997) Common Rail Injection System for Commercial Diesel Vehicles. SAE paper 970345.

[2] Alfuso S, Allocca L, Corcione FE, Di Stasio S (1999) Atomisation mechanisms of an evaporating Common Rail Diesel spray in high pressure environment” - Proceedings 15th Annual Conference on Liquid Atomisation and Spray Systems – ILASS-Europe ‘99, Toulouse (France).

[3] Docquier N (2002) Optical Investigation of the effect of Fuel Jet Wall Impact Position on Soot Emissions in a Single Cylinder Common Rail Direct Injection Diesel Engine. SAE paper 2002-01-0495.
[4] Borthwick RP, Farrell PV (2002) Fuel Injection Spray and Combustion Chamber Wall Impingement in Large Bore Diesel Engines. SAE paper 2002-01-0496.
[5] Allocca L, Corcione FE, Golini S, Papetti F (1993) Assessment of Wall Heat Transfer Model for an Impinging Diesel Spray. ASME National Heat Transfer Conference: "Symposium on Fire and Combustion System", Aug. 8-11, 1993 – Atlanta, GA.
[6] Katsura N, Saito M, Senda J, Fujimoto H (1989) Characteristic of a Diesel Spray Impinging on a Flat Wall. SAE paper 890264
[7] Mohammadi A, Kidoguki Y, Miwa K (2002) Effect of Injection Parameters and Wall- Impingement on Atomization and Gas Entrainment Processes in Diesel Sprays. SAE paper 2002-01-0497.
[8] Kusakabe T, Tsue M, Kadoka T (1994) Visualization of Diesel Spray by Laser Sheet Method. SAE paper 941920.
[9] Borthwick RP, Farrell PV (2002) Fuel Injection Spray and Combustion Chamber Wall Impingement in Large Bore Diesel Engines. SAE Paper 2002-01-0496.
[10] O'Rorke P J, Amsden A A (1987), "The Tab Method for Numerical Calculation of Spray Droplet Break-up", SAE Paper 872089.
[11] Huh K, Gosman A D (1991), "A Phenomenological Model of Diesel Spray Atomisation", Proceedings of The International Conference of Multiphase Flows, 1991, Tsukuba – Japan.
[12] Ibrahim E A, Yang H Q, Przekwas A J (1993), "Modelling of Spray Droplets Deformation and Break-up" AIAA J. Propulsion and Power, Vol. 9, pp. 651 – 654.
[13] Reitz R D (1996), "Computer Modeling of Sprays", Spray Technology Short, May 1996, Pittsburgh, PA.
[14] Arcoumanis C, Gavaises M, French B (1997), "Effect of Fuel Injection Processes on the Structure of Diesel Sprays, SAE Paper 970799.
[15] Patterson M A, Reitz R D (1998), "Modelling the Effects of Fuel Spray Characteristics of Diesel Engine Combustion and Emission", SAE Paper 980131.
[16] Von K Sarre C, Kong S C, Reitz R D (1999), "Modelling the Effects of Injector Nozzle Geometry on Diesel Sprays", SAE Paper 1999 – 01 – 0912.
[17] Bella G, Rotondi R, Corcione F E, Valentino G (1999), "Experimental and Numerical Analysis of a Diesel Spray" ICE 1999.
[18] Rotondi R, Bella G, Grimaldi C, Postrioti L (2001), "Atomisation of High – Pressure Diesel Spray: Experimental Validation of a New Break-up Model", SAE Paper 2001 – 01 – 1070.
[19] De Vita A, Di Angelo L, Allocca L (2002), "Early Injection and Time-Resolved Evolution of a Spray for GDI Engines", 2002 ASME Fluids Engineering Division Summer Meeting, July 14 –18, 2002, Montreal, CANADA.
[20] Allocca L, De Vita A, Di Angelo L (2002), "Experimental Analysis of a CR Diesel Spray", ILASS – Europe 2002, Zaragoza Sept. 9 – 11.
[21] Allocca L, De Vita A, Di Angelo L (2002), "Wall – Impingement Analysis of a Spray from a Common Rail Injection System for Diesel Engines", THIESEL 2002, Valencia Sept 10 – 13.