1. INTRODUCTION

In recent years, the promise of considerable fuel-economy savings and improved performance has prompted a large extent research and development of spark-ignition direct-injection engines [1-3]. These combustion systems offer significant advantages over conventional port-fuel-injection engines, satisfying the conflicting requirements of mixture preparation during high-load (homogeneous stoichiometric/lean) and part-load (stratified overall lean) conditions [4]. The fuel injection system in a gasoline direct injection (GDI) engine is a key component that must be carefully matched with the in-cylinder flow field to provide the desired mixture cloud over the entire operating range of the engine, producing a well-atomised fuel spray [5]. For the efficient combustion of a stratified mixture, stable and compact spray structure is necessary. The combustion system designs for GDI can be divided into three main types, classified according to the relative position of the injector towards the spark plug and the piston crown and according to the mixture preparation approach [6]. The first production solutions have adopted either wall-guided or air-guided concepts, as shown in Fig. 1. The second-generation of GDI engines will likely use a spray-guided concept with a centrally mounted fuel injector spraying along the cylinder axis towards a spark plug with electrodes located near the edge of the spray [7]. There are presently two different mixture preparation principles under development for spray-guided systems based on the multi-hole and the outward-opening pintle nozzles [8]. Research programmes using both experimental techniques and calculations are currently running, in order to gain understanding of the nozzle flow, spray characteristics and the performance and durability of emerging gasoline high-pressure fuel injection systems as well as their application to direct injection spark-ignition engines.

2. NUMERICAL MODEL

In this section the numerical models employed are briefly described, whereas their mathematical formulation...
can be found in [11,12]. The simulation of the continuous phase, which describes the liquid flow inside the fuel injection system as well as the gas motion in case of spray injection, is performed using the GFS (General Fluid Solver) code, developed in [13,14] within their research group. This is a 3-D, transient, turbulent and multi-phase flow solver that can be applied to geometries with moving boundaries. The time-averaged form of the full Navier-Stokes equations describing the continuity, momentum and conservation equations for scalar variables are numerically solved on a collocated curvilinear non-orthogonal numerical grid. The spatial and temporal discretisation method is based on the finite volume approach using second order schemes. The conventional k-ε model is used here to simulate the effects of turbulence. A two-phase cavitation model based on a Eulerian-Lagrangian approach [11] has been employed for the simulation of cavitating flow, in the case this phenomenon takes place. This model assumes that the cavitation phenomena may be described by discrete bubbles, which undergo various physical processes according to a stochastic Monte-Carlo approximation. Similarly, the spray CFD model implemented in order to investigate the dispersion of the fuel droplets further downstream from the injection point, is based on the numerical solution of the Eulerian-Lagrangian multi-phase flow conservation equations describing the local spray structure and its interaction with the surrounding gas. The model assumes quasi-steady inter-phase transport of mass, momentum and energy modelled using empirical correlations. The parcel initial properties are determined by the flow conditions at the exit of the injector nozzle. This provides estimates for the transient flow rate, discharge coefficient and effective area at the nozzle exit flow area, transiently resolved during the injection period, as shown in [14]. The liquid-core fragmentation process at the exit of hole-type injectors is simulated by turbulence-induced and cavitating-induced atomisation models [14]. Following the start of injection, various spray sub-models are implemented for the calculation of the physical processes taking place in the sub-grid time and length scales; these include liquid droplet aerodynamic break-up, turbulent dispersion, vaporisation, drop-to-drop interaction and wall impingement. The droplet drag coefficient, which determines to a large extent the momentum exchange between the gas and the liquid phases, is modelled for different types of particles, taking into account the effects of movement in an evaporating environment, presence of other droplets, internal flow circulation and non-spherical droplet shape. Droplet aerodynamic break-up plays an important role on the predictions of the droplet size population. The model used combines correlations from various literature findings in order to predict the mean diameter, the droplet deformation and the break-up time over a wide range of Weber numbers. Then, the droplet size is randomly sampled from a distribution function, which is calculated using the maximum entropy formalism [14]. Another important aspect taken into account in the model is related to the numerical treatment of the interaction between the two phases achieved through source terms expressing the mass, momentum and energy between the liquid and gas phases. The grid cells can be split dynamically during the numerical solution to smaller ones in the area of interest (local grid refinement), while the interpolation of the continuous phase properties at the parcel location and the distribution of the source terms and the void-fraction is performed on a number of cells found within a pre-defined interaction distance. This methodology has allowed use of computational cells with size of the same order of magnitude to that of the liquid droplets, resulting to better resolution of the recirculation zones formed by the surrounding air during the spray development [12].

3. NOZZLE FLOW & TEST CASES

A prototype 6-hole injector, with symmetric hole arrangement, nominal cone angle of 90° and maximum operating pressure of 200bar has been investigated. Tests have been carried out injecting iso-octane at two injection pressures of 120 and 200bar and at two chamber pressures of 1 and 12bar, under room temperature. The duration of the injection triggering signal was kept constant at 1.5ms.

The flow conditions at the injector hole exit are predicted by the two-phase cavitation flow model, using the numerical grid shown in Fig. 2(a). The volumetric flow rate, as calculated from a 1-D fuel injection system model, is illustrated in Fig. 2(b) for both nominal rail pressures investigated. Predictions of the cavitation volume fraction inside the injection holes is shown if Fig 2(c). At the entrance of the injection holes, the local pressure falls well below the vapour pressure of the liquid, indicating that cavitation is expected to take place in this area. The cavitation model estimates a hole exit effective area of

![Fig. 2: (a) Numerical grid used for flow simulations inside the six-hole injector nozzle, (b) temporal development of volumetric flow-rate for 120 and 200bar nominal rail pressure, estimated from an 1-D fuel injection system model and (c) volume fraction inside the injection holes due to onset and development of cavitation, at 120bar nominal rail pressure.](image-url)
about 90% of the geometric one and a discharge coefficient around 0.7. These estimates have been used as input into the spray model in order to investigate the development of the spray injected into a constant volume chamber. Spray model predictions have been validated against spray images, obtained with a high-resolution CCD camera, and PDA measurement of droplet mean axial and radial velocity components and arithmetic mean diameter, obtained with a 2-D phase-Doppler anemometry system [10]. The PDA measurement points are shown in Fig. 3. As can be seen, these are concentrated on two horizontal planes located at 10 and 30mm below the nozzle exit.

![Fig. 3: PDA measurement grid points.](image)

**Fig. 4:** (a) Half numerical grid of the 60° sector representing 1/6 of the constant volume chamber and (b) fuel spray droplet distribution and air-flow ribbons 1.0ms after ASOI, coloured according to their total velocity magnitude [Nominal rail pressure 200bar, back-pressure 1bar].

Fig. 4(a) shows the numerical grid used for the spray simulations. This is a 60degrees sector with symmetry boundary conditions, representing the 1/6 of the constant volume chamber. Two different grids have been used. The initial grid consists of approximately 100,000 tetrahedral cells, while the second one is refined automatically in the area of spray development and reaching up to 250,000cells at the end of injection. Fig 4(b) shows the predicted spray structure 1.0ms after start of injection (ASOI) and the corresponding induced by the spray air motion, for the 200bar nominal rail pressure case and for injection under atmospheric conditions. As can be seen, there is a recirculation zone formed at the upper part of the spray and extending about half distance between the injection point and the spray tip. A lower strength recirculation zone can be also observed at the inner part spray, located towards the axis of symmetry of the computational domain. The droplet mean diameter very near the nozzle exit, as estimated by the liquid-core atomisation model, decreases slightly with increasing injection pressure, reaching asymptotically a value of around 20µm. Finally, on the same Fig. 4(c), the induced by the injected spray air velocity magnitude can be observed. Induced air-velocity can reach velocities almost 80-90% of those of the injected droplets near the nozzle exit when injecting under atmospheric conditions, but much lower values for injection against increased air pressure and density.

**4. SPRAY MODEL VALIDATION RESULTS**

In this section, the results obtained for the spray development are discussed in more detail. Initially, sample validation cases against the experimental data published in [10] are presented, followed by two examples of spray injection conditions investigating the effect of different nozzle hole arrangement configuration and the effect of fuel vaporisation on the predicted spray structure.

![Fig. 5: Effect of grid dynamic refinement on the spray penetration using a coarse grid consisting of ~100,000 cells and a fine grid of ~250,000 cells.](image)
during the early injection period, lasting up to 0.5ms. Since the automatically refined grid offers better predictions, it has been used for all subsequent cases investigated. A summary of them can be seen in Fig. 6, which presents the model validation against experimental data for the liquid penetration for all four operating conditions investigated. Model predictions are close to the experimental observations, and as expected, they both confirm that spray tip penetration increases with injection pressure and decreases substantially with increasing back pressure.

![Fig. 6: Comparison between model predictions and experimental data of spray tip penetration for two different nominal injection and back pressure conditions.](image)

The comparison between CCD images and computational results 0.5ms after the start of injection is presented in Fig 7. Fig 7(a) corresponds to the lower injection pressure case of 120bar with injection under atmospheric conditions while Fig 7(b) corresponds to the higher injection case of 200bar keeping the same back pressure. Finally, Fig. 7(c) corresponds to the same high rail pressure case but with injection at elevated back pressure of 12bar. It is clear that increasing back pressure results to a significant reduction of the spray penetration. However, the spray shape remains similar, in terms of spray cone and deflection angles, independently of the injection and back pressures used. This characteristic of the multi-hole injector is a clear advantage compared to the pressure-swirl atomiser for spray-guided combustion systems.

Model predictions are compared against PDA experimental data in Figs. 8, 9 and 10 for the temporal variation of the ensemble averaged droplet mean and rms velocity components and the arithmetic mean (AMD) droplet diameter. All results presented here have been obtained on the spray axis. Fig. 8 refers to measurements at 10mm from the nozzle exit while Fig 9 to the same type of measurements and predictions but this time at a distance further down at 30mm from the nozzle hole. In both of them the results presented reveal the effect of injection pressure on droplet size and velocity for injection under atmospheric conditions. As can be seen, increasing injection pressure from 120 to 200bar has a very small effect on droplet size at 10mm from the nozzle hole. However, there is a significant difference in the injection velocity.

![Fig. 7: Comparison between CCD images [10] and model predictions from the multi-hole injector at 0.5ms ASOI, revealing (a) $P_{in}=120$bar, $P_{back}=1$bar, (b) $P_{in}=200$bar, $P_{back}=1$bar and (c) $P_{in}=200$bar, $P_{back}=12$bar.](image)

At 30mm from the injection hole (Fig. 9), the droplet size between the two different injection cases is also different, with the lower injection case exhibiting larger droplet sizes. This is attributed not to the liquid atomisation process but rather to the droplet secondary break-up, which seems to be enhanced with increasing injection pressure. Model predictions suggest that these processes are completed about 15mm from the nozzle exit, and thus, can be only realised at the 30mm measurement plane.

Fig. 10 reveals the effect of increasing back pressure on the predicted droplet characteristics at 30mm below the nozzle exit, for the 120bar nominal rail pressure case. As can be seen, both experimental data and model predictions suggest that increasing the back pressure not only results to significant droplet deceleration due to increased drag, but also to much larger droplets. It seems that the fast droplet deceleration takes place at time scales shorter than those required for droplet aerodynamic break-up, resulting to the
observed and calculated increase in the droplet size far downstream of the injection point.

---

**Fig. 8:** Temporal variation of droplet (a) droplet AMD (b) mean and rms axial velocity component and (c) mean and rms radial velocity component, 10mm below the nozzle exit under atmospheric conditions.

---

**Fig. 9:** Temporal variation of droplet (a) droplet AMD (b) mean and rms axial velocity component and (c) mean and rms radial velocity component, 30mm below the nozzle exit under atmospheric conditions [symbols legend as in Fig. 8].

---

**Fig. 10:** Temporal variation of droplet (a) AMD, (b) mean and rms axial velocity component and (c) mean and rms radial velocity component, for nominal injection pressures of 200bar, chamber pressures of 1 and 12bar, at 30mm below the nozzle exit.

---

5. PARAMETRIC INVESTIGATION

Since model predictions seem to reproduce reasonably well the measured spray characteristics for both injection and back pressure conditions investigated, it was considered useful to use the computational model in order to investigate the effect of parameters related to the nozzle design and operating conditions. Among the simulation cases performed, only a limited number of them will be presented here. These include a simulation case for a ‘6+1’ multi-hole nozzle arrangement and an injection case under increased air temperature, which correspond to injection during the late compression stroke of a GDI engine.

Combination of different hole arrangements and injector positioning relative to the spark plug can offer flexibility in obtaining the desired air-fuel vapour distribution at the time of ignition, since the air motion induced from the interaction between the injected sprays affects the fuel vapour distribution. Fig. 11 demonstrates such an example. It refers to injection from an injector having a central hole in addition to the six side holes. The presence of this central hole aims to produce vapour in the area between the six symmetrically located sprays of the original design. As can be seen, this configuration enhances the formation of a recirculation zone located at the inner part of the side spray. This recirculation zone has been found to be relatively unstable, not only because the central hole spray never develops in a perfectly symmetric way, but mainly because
the cavitation structures of internal flow of the central hole has been found to develop in a more unstable fashion compared to the side holes. As a result, the spray injected from the central hole can be over-penetrating compared to the rest, leading to undesirable wall impingement and also exhibiting significant cycle-to-cycle variations. This is a case that the computational model has provided insight to the reasons leading to the observed undesirable central-hole spray characteristics.

The second example to be presented here refers to a case of spray injection under elevated air temperatures. This can be the case of injection during the compression stroke of a direct injection gasoline engine. Fig. 12(a) and 12(b) show the comparison of the spray structure between the non-evaporating spray previously examined and an evaporating one at 388K. In addition to the air-flow ribbons and the liquid droplets plotted, the fuel vapour mass fraction distribution in plotted for the evaporating case. As can be seen, the iso-octane fuel is vaporising relatively fast, leading to relatively weaker induced air motion recirculation zones. At the same time, the liquid phase seems to penetrate significantly less compared to the non-evaporating one. This can be seen more clearly in Fig. 12(c), which presents the calculated spray penetration for both cases. As can be seen, when evaporation takes place, liquid penetration stops at a distance downstream of the injection hole, although there is no significant difference during the initial part of the spray penetration curve. At this liquid-length distance, the total spray evaporation rate becomes equal to the fuel injection rate. Clearly, this flow characteristics, being a function of the injector geometric characteristics and back chamber thermodynamic conditions, but less dependent on injection pressure, can greatly affect the design of the combustion system and the selection of the appropriate injection nozzle. In Fig. 12(c), in addition to the liquid phase penetration plotted, the vapour phase penetration is also plotted. As can be seen, vapour continues to penetrate even after the liquid phase penetration stops, due to the momentum transfer from the injected liquid to the surrounding air.

5. CONCLUSIONS

Computational and experimental results for sprays generated from multi-hole injectors, recently introduced for spray-guided direct injection gasoline engines, have been presented and discussed. The test cases investigated include injection of n-octane at nominal rail pressures of 120 and 200bar into air under both atmospheric and high pressure back chamber conditions. Model predictions have been validated against phase Doppler anemometry measurements of the droplet size and velocity as well as high resolution CCD spray images, which provide the temporal evolution of liquid penetration and spray shape. Internal nozzle flow calculations have provided information both for the internal injection hole flow characteristics and the subsequent spray development. These results give the necessary information required to investigate the effect of the fuel injection system design on the subsequent spray characteristics. Spray model predictions obtained using as input the predicted flow distribution at the nozzle exit and combination with appropriate atomisation models, have resulted to reasonable predictions of the spray structure for all operating conditions investigated. Within the measured range, the effect of injection pressure on droplet size was rather small while the increase in chamber pressure from atmospheric to 12bar, resulted in much smaller droplet velocities and larger droplet sizes. Overall, the investigation confirmed the advantages of new generation high-pressure multi-hole
injectors for gasoline direct-injection engines, compared to swirl pressure atomizers, in terms of spray structure stability under varying chamber thermodynamic and injection operating conditions.

REFERENCES


