A numerical characterization of new high-pressure multi-hole GDI injector

S. Malaguti*, S. Fontanesi, G. Cantore
* Dipartimento di Ingegneria Meccanica e Civile
Università degli Studi di Modena e Reggio Emilia
Modena, 41125 Italy

Abstract

The paper reports a numerical activity on the investigation of the spray structure in a new-generation GDI injector. The spray is investigated under quiescent conditions, injecting the fuel in a test vessel under non-evaporative ambient conditions.

Results from 3D-CFD simulations are compared to experimental measurements available in literature: commercial gasoline at two different injection pressures (10 and 20 MPa) was injected and the spray evolution was analyzed throughout the injection duration.

The spray was investigated along the jet axis by the phase Doppler anemometry in order to provide the size and two components of the droplets velocity: axial and radial. Data were analyzed using the ensemble averaging technique in order to provide mean values.

Experimental measurements briefly described above are used to test and validate some lagrangian spray numerical sub-models and numerical parameters such as grid density, numerical setup, primary and secondary fuel atomization and droplet to droplet interaction. Particular care is devoted to the accurate representation of spray primary breakup, in view of the lack of ad-hoc developed models available in literature. A wide CFD activity is therefore performed in order to investigate grid effects on the prediction of liquid spray penetration and droplet velocity.

Results from the CFD analyses show a relevant dependency of the spray structure on both the computational cell size and the adopted CFD model ensemble.

Introduction

Second generation of high-pressure GDI injectors are characterized by a multi-hole structure aiming at maximizing fuel-air mixing and fuel vapor diffusion in the combustion chamber. Figure 1 shows the 6-hole GDI injector investigated in the present paper. The specific hole spatial distribution and orientation produces a strong jet to jet interaction, as visible in Figure 2. Due to the relatively low overall cone angle (< 90 degrees) the resulting spray structure appears to be narrow and single jets are not clearly distinguishable.

The above described particular spray structure is requested to meet a proper fuel-air mixing over a wide range of injection pressures and engine operating conditions.

In view of both the particular injector configuration and the lack of either experimental and numerical data available in literature, a wide CFD campaign is carried out in order to investigate the spray behavior in terms of both pointwise and global parameters.

Results and Discussion

Different primary break-up models available in literature are at first tested and evacuate by means of a 3D-CFD simulations, within the framework of the lagrangian approach to the modeling of fuel sprays.

Even if the above process is carried out iteratively in combination with a wide activity on the tuning of droplet secondary break-up Reitz-Diwakar model's constants, both Wave and Huh-Gosman primary break-up models show a relevant inadequacy in correctly capturing the global spray behavior.

Figures 3 and 4, showing droplet average diameter D10 and droplets velocity respectively, highlight the above statements.

The sensitivity analysis on droplet break-up is then extended to some relevant CFD parameters, such as grid size and orientation, which are widely known to play a relevant role on the estimation of droplet drag coefficient. In view of the spatial complexity of the actual injector, results from the full 6-hole injector are compared to those from a simplified model based on a single hole.

As a final stage, a user-implemented routine is tested aiming at manually specifying droplet initial conditions at the injector holes, therefore avoiding primary break-up model deficiencies.

* Corresponding author: simone.malaguti@unimore.it

Different droplet distributions are analyzed, showing promising results in terms of both pointwise and global spray parameters. Results are also useful to highlight the responsibility of different physical phenomena on the above mentioned deficiencies of the tested primary break-up models for the particular class of GDI injectors.

Cavitation, jet to jet interaction and more robust droplet-droplet collision models would be of crucial importance to increase the predictive capabilities of the CFD models.

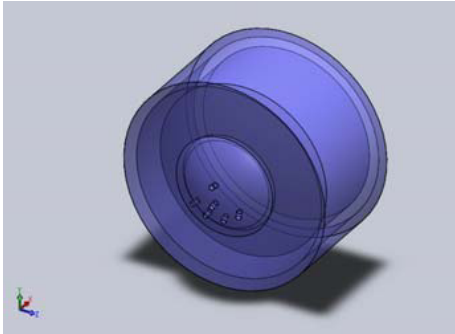

Figure 1. The Investigated injector 6-hole nozzle sketch

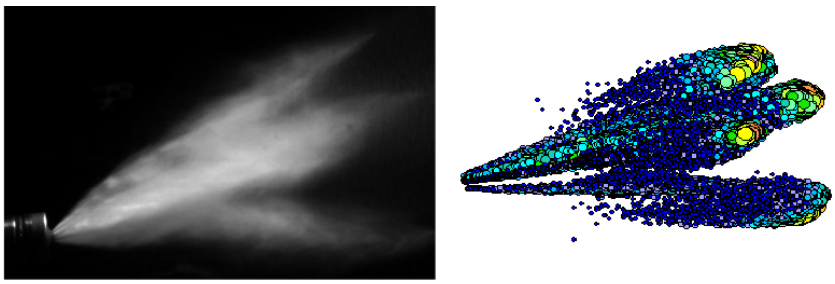

Figure 2. Experimental and numerical comparison spray image

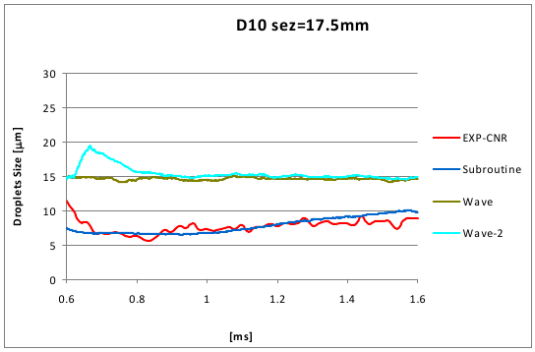

Figure 3. Droplet average diameter comparisons @ 17.5 mm from injector nozzle

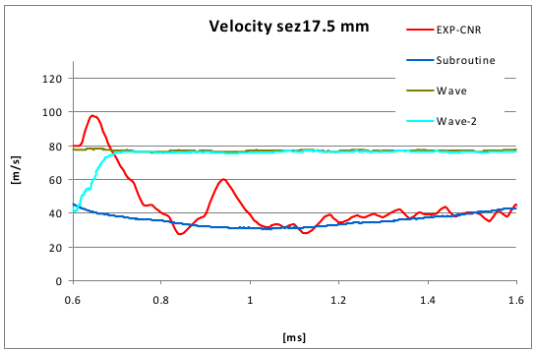

Figure 4. Droplet axial velocity comparisons @ 17.5 mm from injector nozzle