

Numerical Simulation of the Effect of 3D Needle Movement on Cavitation and Spray Formation in a Diesel Injector

B Mandumpala Devassy¹, W Edelbauer¹, D Greif²

¹AVL List GmbH, Hans List Platz 1, 8020 Graz, Austria

²AVL-AST d.o.o., Ul. kneza Koclja 22, 2000 Maribor, Slovenia

E-mail: Bejoy.MandumpalaDevassy@avl.com

Abstract. Cavitation and its effect on spray formation and its dispersion play a crucial role in proper engine combustion and controlled emission. This study focuses on these effects in a typical common rail 6-hole diesel injector accounting for 3D needle movement and flow compressibility effects. Coupled numerical simulations using 1D and 3D CFD codes are used for this investigation. Previous studies in this direction have already presented a detailed structure of the adopted methodology. Compared to the previous analysis, the present study investigates the effect of 3D needle movement and cavitation on the spray formation for pilot and main injection events for a typical diesel engine operating point. The present setup performs a 3D compressible multiphase simulation coupled with a standalone 1D high pressure flow simulation. The simulation proceeds by the mutual communication between 1D and 3D solvers. In this work a typical common rail injector with a mini-sac nozzle is studied. The lateral and radial movement of the needle and its effect on the cavitation generation and the subsequent spray penetration are analyzed. The result indicates the effect of compressibility of the liquid on damping the needle forces, and also the difference in the spray penetration levels due to the asymmetrical flow field. Therefore, this work intends to provide an efficient and user-friendly engineering tool for simulating a complete fuel injector including spray propagation.

1. Introduction

Fuel injectors are one of the essential components in an internal combustion engine system. High pressure injection systems employ adequate atomization and mixture formation which are required for improved engine performance and reduced emissions. The present work addresses high pressure injection events including fuel injection and sprays by using a coupled 1D/3D nozzle flow simulation technique. The method has already been discussed in previous studies, as shown in [1] and [2]. The present approach introduces the co-simulation technique for pilot and main injections, and it is extended in such a way that the 3D motion of the needle, with 3 degrees-of-freedom, can be simulated in the 1D code: AVL BOOST- HYDSIM [3], and the corresponding mesh movement is represented in the 3D code: AVL FIRE® [4]. The longitudinal and transverse motion of the needle is coupled with the cavitating 3D nozzle flow. At selected time instants the flow field at the nozzle exit is written to separate data files, called nozzle-files. These files provide the boundary conditions for the sub-sequent Euler-Lagrangian spray simulation of the combustion chamber. The high pressure liquid medium is treated compressible [1], and the effect of needle damping forces is studied thoroughly. Cavitation and spray models have already been described in previous works, and the main approaches are based on the Euler-Eulerian, Euler-Lagrangian formulation, or their combination, as discussed in [5] and [6] for



the spray and in [7] and [8] for the cavitation and cavitation erosion models. The authors would like to refer to their previous works in [1] for detailing the applied 1D/2D/3D coupling interface, the governing equations and different models employed for the current simulation. The present study intends to investigate the spray injected from an eroded-asymmetrical and a non-eroded-symmetrical nozzle, which is already described in [1].

The paper is organized as follows: the second section presents the details of the test case with the computational mesh, and initial and boundary conditions. Section 3 provides a brief analysis of the results with a detailed discussion, and Section 4 contains conclusions and discusses future work.

2. Six-hole Fuel Injection Test Setup

The fuel injector with the combustion chamber geometry is shown in Figure 1. The motivation of this study is to identify the effect of 3D needle movement, compressibility and nozzle asymmetry on the spray formation and its characteristics. Figure 1 illustrates the mesh of the eroded 6-hole injector in a zoom-in view, and the piston bowl geometry of the combustion chamber. Typical diesel engine conditions at a full-load operation point, as listed in Table 1, are applied for the spray simulation. The computational mesh of the injector flow simulation contains the nozzle sac, the needle shaft, the needle seat and the nozzle holes. Here, the asymmetric geometry is supposed to mimic the production inconsistency and/or possible erosion effects. For the actual injector the eroded regions in the nozzle holes have been randomly chosen. The full needle movement with three degrees of freedom is studied. Consequently a full 360 degree geometry is necessary for the current investigation.

In the present study the 3D-1D coupled simulation runs for 57 hrs using 16 Intel(R) Xeon(R) CPU X5620 @ 2.40GHz processors.

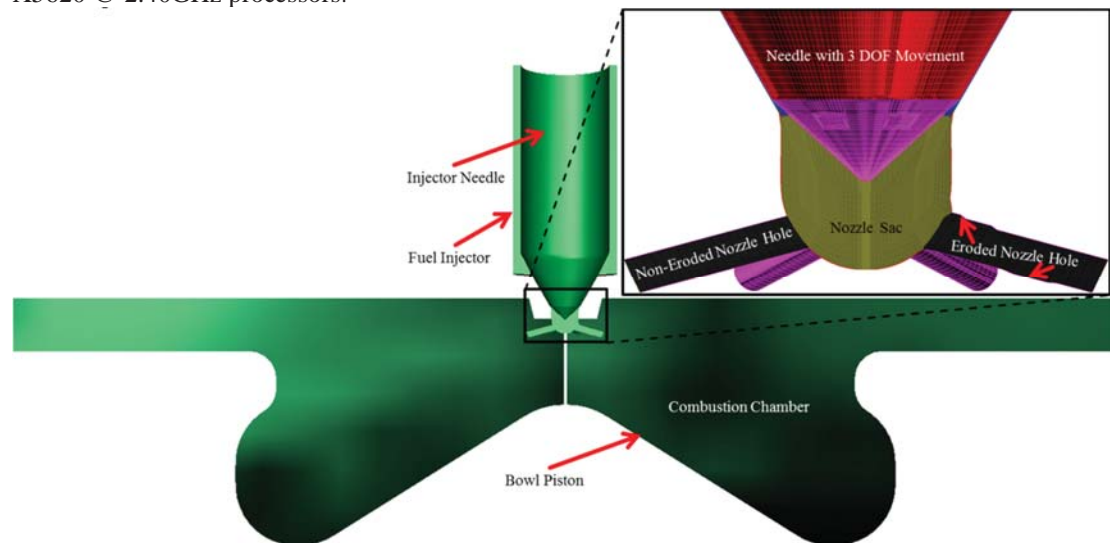


Figure 1. Cross-sectional view of the eroded 6-hole fuel injector with combustion chamber.

Table 1. Operating Conditions.

Liquid density	830 kg/m ³
Temperature of the fuel inlet	333.15 K
Each nozzle hole diameter, D	218.5 μm
Injection pressure	160 MPa
Pressure in the combustion chamber	6.7 MPa
Temperature in the combustion chamber	900 K
Piston, head and liner wall boundaries	550, 500 & 400 K

Table 1 shows the operating conditions of the simulation. The calculation is carried out for 2.3 ms with a rail pressure of 160 MPa. Pilot and main injection events are taken into account and cavitation as well as spray formation is studied for the symmetrical and the asymmetrical nozzle condition.

3. Results and Discussion

For both cases a considerable quantity of vapor has been detected. For the symmetrical mesh, the vapor volume fraction appears to be symmetric, while for the asymmetrical mesh, owing to the eroded nozzle-hole regions, the vapor regions are clearly different from hole to hole. Note, that different holes exhibit different cavitation patterns. In several holes the cavitation cloud reaches the nozzle orifice. Figure 2 and Figure 3 present the liquid mass flow rates, and the cumulative quantity of liquid fuel mass injected for the entire injection cycle. The difference in the curves is caused by the differences in the vapor generation, where symmetrical geometry produces more vapor than the asymmetrical geometry. This behavior is also visible in the zoom-in view in Figure 4.

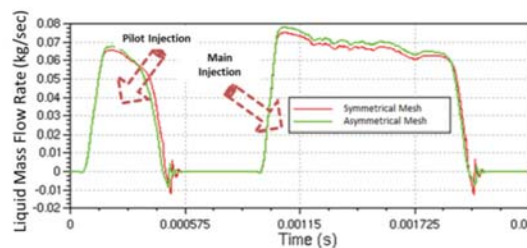


Figure 2. Liquid mass flow rate for symmetrical and asymmetrical geometries.

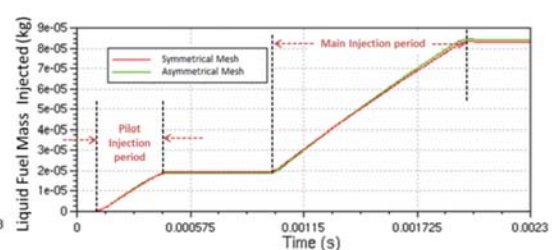


Figure 3. Liquid and vapor fuel mass injected for symmetrical and asymmetrical geometries.

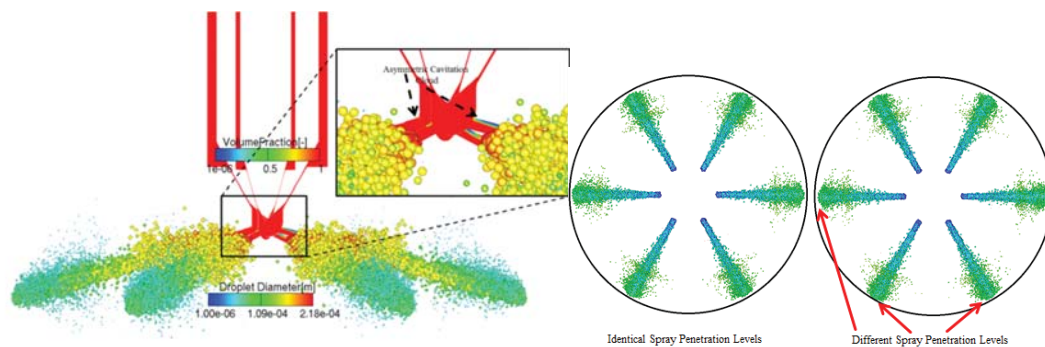


Figure 4. Left: Fuel injection and sprays from an eroded 6-hole asymmetrical nozzle at 0.16ms after start of injection. Right: Different spray penetration levels from a six-hole symmetrical (left) and asymmetrical nozzle holes (right).

In the applied workflow two separate calculations, using the Eulerian approach as a two-fluid model inside the nozzle tip, and the Lagrangian Discrete Droplet Model (DDM) approach outside the injector in the spray regions, are performed. The flow data in the nozzle orifice are transferred to the DDM model [5] as initial conditions. Additionally, a primary break-up model is applied within the DDM. The results show that the asymmetrical flow field generated from an eroded nozzle (also shown in Figure 4) develops an asymmetrical spray mainly due to the variation in cavitation fields and the oscillatory needle movement. This is clearly notable by visualizing the spray characteristics, especially the spray tip penetration, as shown in Figure 4 (right). There one can observe slight differences in the spray contour and clear differences in the spray tip penetration between eroded and non-eroded nozzle

holes. Figure 5 (left) gives detailed insight of the eroded nozzle-hole positions. The induced asymmetry here is supposed to mimic the production inconsistency and the erosion effects. For the actual injector the eroded regions in the nozzle holes have been randomly chosen; denoted as 'E' in Figure 5 (left). Due to the difference in the flow field (see Figure 4), the injected fuel mass varies between the nozzle holes, as shown in the bar chart of Figure 5 (right). This difference is mainly due to the uneven distribution of cavitation regions causing the difference in the spray penetration length.

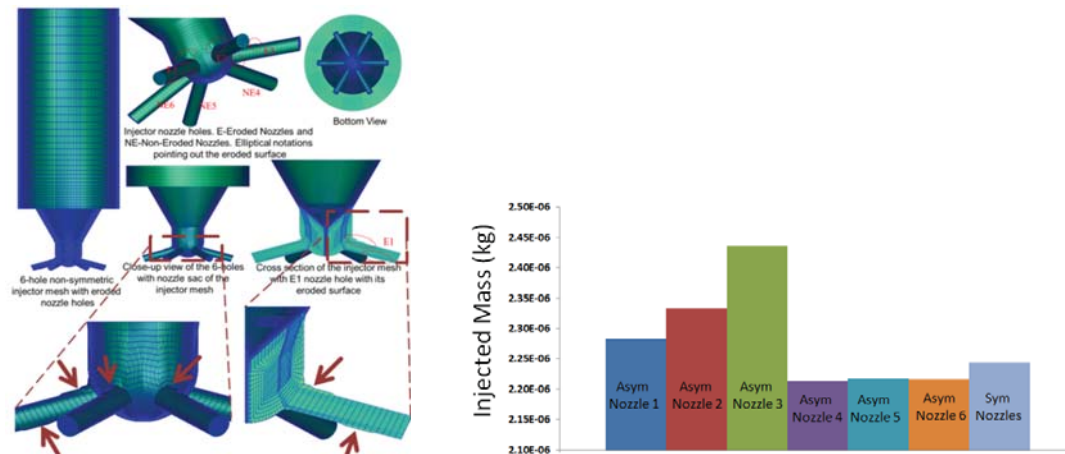


Figure 5. Left: Asymmetrical 6-hole injector computational mesh. The asymmetry is due to the eroded nozzle holes sketched as “E”. Right: Comparison of the injected fuel mass for different nozzle holes of the asymmetric and the symmetric injector nozzle holes until 0.3ms after start of the pilot injection.

4. Conclusions

This work intends to provide a novel flow simulation technique for studying nozzle cavitation and spray formation for pilot and main injection events in a multi-hole fuel injector. The results indicate clear differences in terms of cavitation generation and flow fields for both, the symmetric and the asymmetric, test case geometries. Owing to the asymmetrical flow field in the injector, differences in the spray tip penetration are clearly visible. This study combines AVL FIRE® and AVL BOOST-HYDSIM tools in order to simulate the flow in a high-pressure fuel injector and the subsequent spray formation. Liquid compressibility, cavitation and 3D needle movement are taken into account.

References

- [1] Mandumpala Devassy B, Caika V, Sampl P, Edelbauer W, and Greif D 2015 *Proc. Institution of Mechanical Engineers Conference on Fuel Systems for IC Engines (London, March 10-11)*.
- [2] Caika V, Sampl P, and Greif D 2013 Integrated 1D/2D/3D Simulation of Fuel Injection and Nozzle Cavitation *SAE Int. J. Engines* **6** (3) (doi:10.4271/2013-24-0006).
- [3] AVL BOOST HYDSIM 2013 User's Guide, Graz.
- [4] AVL FIRE®, Eulerian Multiphase 2014 Graz.
- [5] Von Berg E, Edelbauer W, Alajbegovic A, Tatschl R, Volmajer M, Kegl B and Ganippa L C 2005 Coupled Simulations of Nozzle Flow, Primary Fuel Jet Breakup, and Spray Formation *Journal of Engineering for Gas Turbines and Power* **127** (4) (ASME, New York) pp 897-907.
- [6] Edelbauer W 2014 Coupling of 3D Eulerian and Lagrangian Spray Approaches in Industrial Combustion Engine Simulation *Journal of Energy and Power Engineering* **8** pp 190-200.
- [7] Alajbegovic A, Grogger H and Phillip H 1999 Calculation of Transient Cavitation in Nozzle using Two-Fluid Method *Int. Conf. on Liquid Atomization and Spray Systems* (Indianapolis, IN, USA, 209-213).
- [8] Greif D and Wang D M 2007 Modeling of Cavitation Erosion within Diesel Injector and its Effect on Spray Propagation *Int. Conference Towards Clean Diesel Engines* (Ischia, Italy).