Large Eddy Simulation of diesel injector opening with a two phase cavitation model

P Koukouvinis[1,2], M Gavaises[2], J Li[3], L Wang[3]

[2] City University London, Northampton Square EC1V 0HB, United Kingdom
[3] Caterpillar Inc., Peoria, Illinois Area US, 61602US

foivos.koukouvinis.1@city.ac.uk

Abstract. In the current paper, indicative results of the flow simulation during the opening phase of a Diesel injector are presented. In order to capture the complex flow field and cavitation structures forming in the injector, Large Eddy Simulation has been employed, whereas compressibility of the liquid was included. For taking into account cavitation effects, a two phase homogenous mixture model was employed. The mass transfer rate of the mixture model was adjusted to limit as much as possible the occurrence of negative pressures. During the simulation, pressure peaks have been found in areas of vapour collapse, with magnitude beyond 4000bar, which is higher that the yield stress of common materials. The locations of such pressure peaks corresponds well with the actual erosion location as found from X ray scans.

1. Introduction

Diesel injection systems play a fundamental role in internal combustion engines since they affect the formation of the fuel spray, atomization and combustion, the formed emissions and the engine efficiency. The jet velocities formed are of the order of 500m/s, requiring very high upstream pressures around 2000bar or even more. Current trends show injection pressures to even rise to 3000bar, in order to meet the future EU legislations in emissions. However, moving towards higher and higher pressure levels causes very high velocities in the tight passages in the Diesel injector and strong accelerations in sharp direction changes (corners, fillets etc.), leading to local static pressure drop below the saturation pressure and causing cavitation. Furthermore, cavitation collapse may lead to cavitation erosion damage and serious degradation of the injector performance, even leading to catastrophic injector failure, which could damage the engine, if the injector tip breaks off.

There have been many efforts for the prediction of the Diesel injector erosion, see e.g. the work of Koukouvinis et al. [1]. The aim of the current work is to simulate the flow inside a Diesel injector in a more fundamental level, including compressibility effects of the liquid phase and also using a Large Eddy Simulation for describing turbulence. The reason for employing compressibility effects is the large density difference in the injector, which may reach 10%, not to mention the high liquid velocities that can reach a Mach number of 0.5. Furthermore, resorting to Large Eddy Simulation techniques is because RANS/URANS are inadequate for capturing the complicate vortex patterns which affect cavitation formation [2], while even modified RANS turbulence models are situational [3]. To the

---

[1] Corresponding author.

authors knowledge, a similar work does not exist in literature; Sezal et al. [4] and Salvador et al. [5] include compressibility effects, but no needle motion.

2. Numerical background

Numerical simulations presented in this work are based on a the solution of the Navier Stokes equations, using a commercial pressure-based solver, Fluent [6]. The equations solved consist of the continuity and momentum equations, while the energy equation has been omitted since heat effects were ignored. However, since Diesel properties vary significantly with the pressure levels in the injection systems, both liquid phase viscosity and density are assumed variable, as functions of pressure. For density, the Tait equation of state was used:

$$p = B\left[\left(\frac{\rho}{\rho_{sat,L}}\right)^{n} - 1\right] + p_{sat} \tag{1}$$

where $\rho_{sat,L}$= 747.65kg/m^3 is the density at saturation pressure p_{sat}=1.2bar, at temperature of 395K. The temperature was estimated through simplified 1D analysis for the pressure levels in the injector [7] and is an estimated average during the injection event. The bulk modulus is B=110MPa and n=7.15.
For Diesel liquid kinematic viscosity, v, a relation provided by N. Kolev [8] is used, applied for the same temperature level as above:

$$\log_{10}(10^6 v) = 0.035065275 - 0.000234373 \cdot p/10^5 \tag{2}$$

For the introduction of cavitation effects, an additional transport equation is solved for tracking the vapour phase. The vapour properties used, are density ρ_V=6.5kg/m3, the dynamic viscosity μ_v=7.5μPa.s. The two phase model, assumes mechanical equilibrium between the two phases, i.e. both liquid and vapour phase share the same pressure and velocity fields. An additional advection equation corresponding to the vapour fraction is solved, in the following form:

$$\frac{\partial(a\rho_v)}{\partial t} + \nabla(a\rho_v \mathbf{u}) = R_e - R_c \tag{1}$$

where a is the vapour fraction, ρ_v is the vapour density, $\mathbf{u}$ is the velocity field and R_e, R_c are the mass transfer rates for condensation (c) and evaporation (e), prescribed by the Zwart-Gerber-Belamri model [9]. The mass transfer terms have been tuned in order to limit the existence of negative pressures inside the computational domain as much as possible; after the tuning the minimum pressure inside the injector is approximately -20bar.
For the capturing of the complicated turbulent structures inside the sac volume and orifice, the Wall Adapted Local Eddy-viscosity (WALE) LES model was employed [10], since it has much better performance in wall-bounded flows, as the one developing inside the injector.

3. Diesel injector simulation

3.1. Case set-up

The geometry has been provided by Caterpillar Inc.; it is a 5-hole tapered injector (see fig. 1a). The injector operates at an average inlet pressure level of 1800bar and outlet pressure of 50bar. The exact pressure pulse and the corresponding needle motion were provided by Caterpillar, calculated using 1D CFD tools, and were used for the simulation. Note also that at the end of the orifice of the injector an additional hemispherical volume was added (fig. 1b), in order to move the influence of the outlet boundary further away from the orifice, especially considering that cavitation structures may reach or even exit the orifice, as will be shown later. The configuration resembles the injection test benches in Caterpillar, where injection is performed in a chamber filled with liquid.
The needle motion is assumed to be in the axial direction only, so any eccentricity effects were omitted. In reality eccentricity effects are significant during the early opening and late closing phases, but introduction of such effects will impose the simulation of the complete injector geometry, which will be much more complicated. In the current simulation, only 1/5th of the domain was solved and

periodic boundary conditions have been employed at the sides of the domain. The computational domain was split in a set of moving, deforming and stationary zones, as shown in fig. 1b.

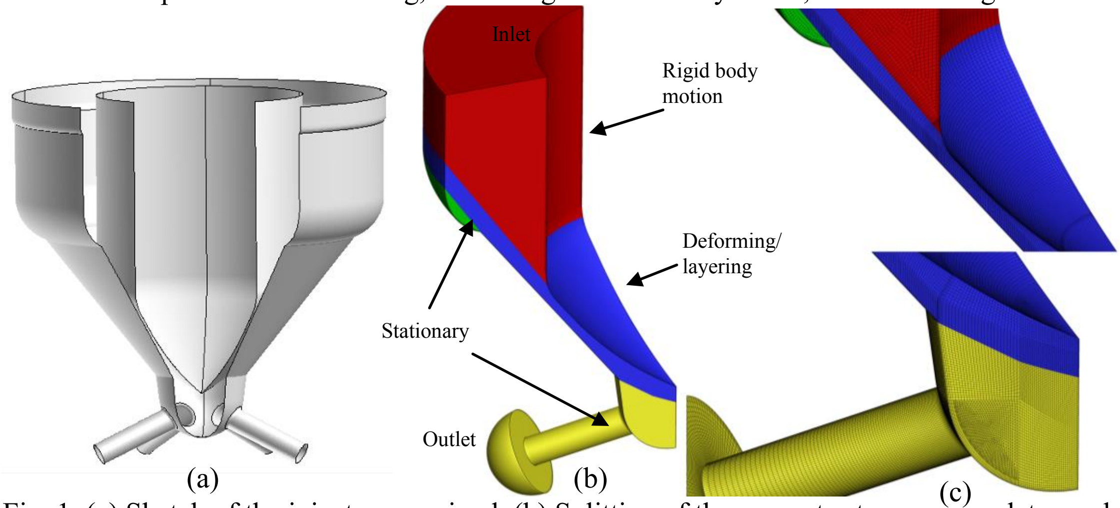

Fig. 1. (a) Sketch of the injector examined, (b) Splitting of the geometry to accommodate mesh motion (c) details of the mesh at the needle seat passage and sac volume.

The computational mesh used is block-structured, with the exception of a zone in the sac prior to the orifice entrance, which is unstructured tetrahedral (see fig. 1c). Inflation layers were introduced in the area of interest, such as sac and orifice to capture the boundary layers. Mesh motion was introduced by a smoothing algorithm at low lifts and the mesh layer addition at higher lifts. The mesh size was approximately 1 million cells at the beginning of the simulation; the needle motion introduces additional cell layers inside the volume denoted with blue colour in fig. 1b, so the actual cell count increases over time. A bounded central scheme (hybrid between central and second order upwind) was used for momentum, while second order upwind for density and QUICK for volume fraction. Time advancement was performed with an implicit, second order, backward differentiation with a time step of 5ns, in order to be able to capture the complicated turbulent patterns. The implicit integration avoids the time step restriction due to compressibility effects, which would further limit the time step to even lower values.

3.2. Simulation results

In fig. 2a the complicated cavitation patterns inside the sac volume and the injector orifice are shown. Also in fig. 2b the evolution of the maximum pressure on the wall is shown. From the simulation results, cavitation starts to develop at the edges of the jet formed at low lifts between the needle and needle seat, due to shear layer instability. Later on, cavitation extends inside the sac volume, due to the vortex formation. At the same time, cavitation collapses on the needle produce strong shock waves, with pressure levels exceeding 4000bar and even peaking at 10000bar. When the needle opening is high enough, e.g. at ~39μm pressure starts to rise inside the sac volume and cavitation structures collapse, causing sporadic pressure peaks on the sac wall. Eventually, cavitation is limited only inside the orifice, at the upper orifice surface. Note that cavitation may extend outside the injector orifice.

4. Conclusion

This paper outlines the methodology and results from Large Eddy Simulation inside a Diesel injector, including liquid compressibility effects. The turbulence model used is capable of capturing the complicated vortex patterns inside the injector sac and orifice, which in turn affect the cavitation formation location and collapse. Cavitation collapses produce strong shockwaves, with pressure levels exceeding 4000bar. The locations of the pressure peaks correspond well with erosion patterns from X-ray imaging of the actual tested injectors.

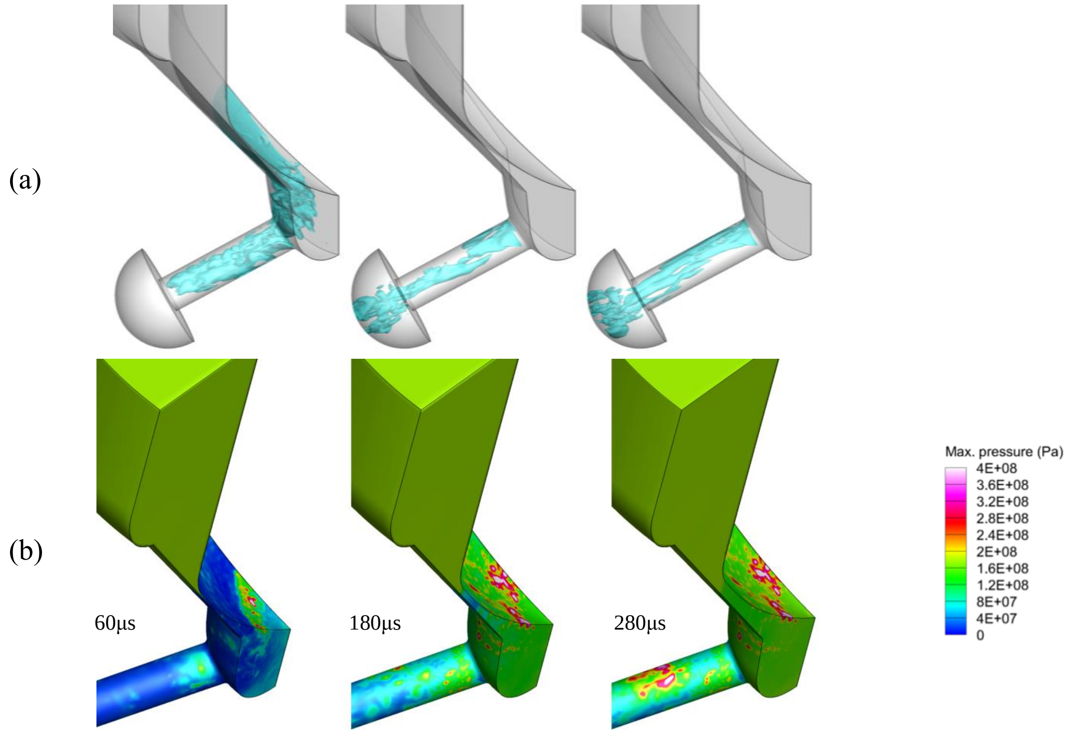

Fig. 2. (a) Isosurface of the vapour volume fraction (b) Accumulation of maximum pressure peaks on the injector walls for three time instances.

Acknowledgements
The research leading to these results has received funding from the People Programme (IAPP Marie Curie Actions) of the European Union's Seventh Framework Programme FP7/2007-2013/ under REA grant agreement n. 324313.

References

[1] Koukouvinis P, et al. 2014 Simulation of cavitation inside diesel injectors, including erosion modelling, in *Fuel Systems for IC Engines*Institution of Mechanical Engineers (IMechE, London). p. 125-135.
[2] Coutier-Delgosha O, Reboud J L, and Delannoy Y 2003 Numerical simulation of the unsteady behaviour of cavitating flows. *Int J Numer Methods Fluids*, **42**: p. 527-548.
[3] Edelbauer W, Strucl J, and Morozov A 2014 Large Eddy Simulation of cavitating throttle flow, in *SimHydro 2014:Modelling of rapid transitory flows* (Sophia Antipolis).
[4] Sezal I H, et al. 2009 Shock and wave dynamics in cavitating compressible liquid flows
in injection nozzles. *Shock Waves* **19**: p. 49-58.
[5] Salvador F J, et al. 2013 Computational study of the cavitation phenomenon and its interaction with the turbulence developed in diesel injector nozzles by Large Eddy Simulation (LES). *Mathematical and Computer Modelling*, **57**: p. 1656-1662.
[6] ANSYS 2013 ANSYS Fluent 15.07.
[7] Strotos G, et al. 2015 Transient heating effects in high pressure Diesel injector nozzles. *Int J Heat Fluid Flow*, **51**: p. 257-267.
[8] Kolev N 2007 Multiphase Flow Dynamics 3. (Springer Berlin Heidelberg).
[9] Ji B, et al. 2013 Numerical analysis of unsteady cavitating turbulent flow and shedding horse-shoe vortex structure around a twisted hydrofoil. *Int J Multiphase Flow*, **51**: p. 33-43.
[10] Nicoud F and Ducros F 1999 Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion*, **62**: p. 183-200.