10th International Symposium on Cavitation - CAV2018
Baltimore, Maryland, USA, May 14 – 16, 2018

Internal and near nozzle flow simulations of gasoline multi-hole injector (ECN Spray G) with transient needle motion

[1]Balaji Mohan*; [1]Mohammed Jaasim; [1]Francisco Hernandez Perez; [2]Jaeheon Sim; [1]William Roberts; [1]Hong Im

[1]King Abdullah University of Science and Technology, Thuwal, KSA; [2]Saudi Aramco, Dhahran, KSA

Abstract

In this work, the effect of transient needle motion on gasoline multi-hole injector's internal and near nozzle flow was studied. Spray G nozzle, an eight hole counter-bore injector from the Engine Combustion Network (ECN), was considered as the fluid domain. Simulations considered the effect of turbulence, cavitation, flash-boiling, compressibility and non-condensable gases under transient needle lift. To model the two phase flow inside and outside the nozzle, the Homogeneous Relaxation Model (HRM) coupled with the Volume of Fluid (VOF) approach was used. HRM model is used in this study because it uses an empirical timescale to reproduce a range of vaporizations mechanisms i.e. both cavitation and flash boiling mechanisms. To model turbulence, the RNG $k - \varepsilon$ model was used. Simulations were performed with two different boundary conditions for the outlet domain to investigate non-flashing and evaporative (Spray-G) and flashing (Spray-G2) conditions. The simulation results were qualitatively validated against the experimental images and quantitatively against the experimental rate of injection (ROI) profile. The liquid plume angle showed good agreement with that of experimental measurement. The results show that the simulation is capable of capturing the ROI accurately with upstream pressure boundary condition and transient needle lift profile. Furthermore, the results show that the hole-to-hole variation in the total injected mass was not very significant. The simulation was also able to capture the cavitation phenomena inside the nozzles and flash boiling in the near nozzle region.

Keywords: Spray G injector, flashing, non-flashing, rate of injection (ROI)

Introduction

Gasoline compression ignition (GCI) engines are becoming popular to exploit the advantage of higher fuel volatility and potentially lower aromatic content [1]. To improve the fidelity of simulation-based GCI combustion system development, it is mandatory to enhance the prediction of injection characteristics. Fuel injection always involves phase-change phenomena. The two most common phase-change phenomena are cavitation and flash boiling. The phase-change process is highly complex in nature. Cavitation takes place in high pressure diesel injectors and is a pressure-driven vaporization taking place at low temperatures. At high fuel temperatures, more energy is required for phase-transition per unit volume of vapor due to high saturation vapor density compared to that of low temepratu re fuel. Thus, flash boiling is a thermal non-equilibirium process unlike cavitation. The non-dimensional Jacob number ($Ja = \frac{\rho_l C_P \Delta T}{\rho_v h_{l_v}}$) which is the ratio of sensible heat energy available to the energy required for vaporization can be used to explain this phenomenon [2]. The high Jacob number indicates that there is abundance of energy available in the liquid to generate vapor. This means that the process is close to equilibirium since heat transfer time-scale is much lower than flow time-scale, and hence cavitating. Alternatively, when the Jacob number is low, the process is non-equilibirium, i.e. flash boiling and the heat transfer time scales will be of the same order as that of the flow time scales [3].

There are several numerical studies on flash boiling for GDI injectors available in the literature [3,4,5,6,7]. Moulai et al. [6] used the OpenFOAM® CFD package to simulate the flash boiling condition by using a velocity boundary condition calculated from the rate of injection (ROI) data from experiments. Recently, Baldwin et al. [7] used the OpenFOAM® package to simulate the flash boiling under transient needle motion. Other simulation works have used

*Corresponding Author, Balaji Mohan: balaji.mohan@kaust.edu.sa

the CONVERGE CFD code and successfully demonstrated its potential to predict flash boiling conditions either at fully opened condition or at different static needle lift positions [3,4,5]. In this work, the CONVERGE CFD package is employed to study the flash boiling phenomenon with transient needle motion. In what follows, the numerical model is described and results are presented and discussed.

Numerical methodology

The simulations were performed using the CONVERGE v2.4 CFD package. CONVERGE solves the conservation equations for mass, momentum, and energy, with the addition of a turbulence closure model. To model turbulence, the Reynolds Averaged Navier-Stokes (RANS) approach was employed and, in particular, the $k - \varepsilon$ model was used.

The Volume of Fluid (VOF) approach is used to model the two-phase flow. In the VOF method, a function α is used to represent the void function which is defined as follows

- $\alpha = 0$: the cell contains only liquid
- $0 < \alpha < 1$: the cell contains both liquid and gas
- $\alpha = 1$: the cell contains only gas

The density in the cell is computed by the following equation

$$\rho = \alpha\rho_g + (1 - \alpha)\rho_l \tag{1}$$

where, ρ_g is the gas density and ρ_l is the liquid density.

The void fraction is solved by the following conservation equation

$$\frac{\partial \alpha}{\partial t} + u_i \frac{\partial \alpha}{\partial x_i} = S \tag{2}$$

However, the void fraction is not directly transported as in Eq. (2), instead the species are first solved using the species transport equation

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial \rho_m u_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\rho D \frac{\partial Y_m}{\partial x_j} + S_m\right) \tag{3}$$

where, $\rho_m = Y_m \rho$

The void fraction is then computed based on the liquid and gas mass fractions as follows

$$\alpha = \frac{m_g/\rho_g}{m_g/\rho_g + m_l/\rho_l} \tag{4}$$

where, m_g is the gas mass fraction (i.e. fuel vapor and dissolved gas) and m_l is the liquid mass fraction.

The Homogenous Relaxation Model (HRM) [8] approach is used to evaluate the source term S_m in the species conservation Eq. (3). This approach describes the rate at which the instantaneous mass fraction of vapor in a two-phase mixture will approach its equilibirium. A simple linearized form for this rate was proposed by Bilicki and Kestin [9] as

$$\frac{Dx}{Dt} = \frac{\bar{x} - x}{\theta} \tag{5}$$

where, $\frac{Dx}{Dt}$ is the rate of change of local vapor quality which gives the estimate for S_m, x is instantaneous mass, $\bar{x}$ is equilibrium mass and θ is the time scale over which x reaches to $\bar{x}$. For evaporation, the time scale θ is calculated using the following equation

$$\theta = \theta_0 \alpha^a \varphi^b \tag{6}$$

where, θ_0 is $3.84 * 10^{-7}[s]$, α is the fuel void fraction, $a = -0.54$, $b = -1.76$, and φ is a dimensionless pressure defined as

$$\varphi = \left| \frac{P_{sat} - P}{P_{cr} - P_{sat}} \right| \tag{7}$$

where, P_{sat} is the pressure at saturation temperature and P_{cr} is the critical pressure

Computational domain

The computational domain used in this current study is the internal nozzle geometry of spray G injector from Engine Combustion Network (ECN) [10]. The nozzle has 8 counter-bored holes with 5 dimples as shown in Fig. 1. In order to capture the near nozzle flow, a hemispherical plenum is necessary to extend the outlet domain. In this study, a hemispherical plenum of 9 mm diameter is used based on the studies done by Saha et al. [3,4]. The vertical cut-plane showing the mesh with the 9 mm outlet domain is shown in Fig. 2. A base cellsize of 150 μm is used in this study. Fixed embedding has been used near the walls, inside the holes and inside the hemispherical outlet domain near the nozzles to capture the sharp gradients in velocity, temperature, species, etc. Three levels of fixed embedding have been used. Therefore, the smallest cell size is 8 times smaller than the base grid, i.e., smallest cell size is 150 $\mu m * 2^{-3} =$ 18.75 μm. According to Moulai et al. [5], reasonable predictions were feasible with even a 22.5 μm of minimum grid size.

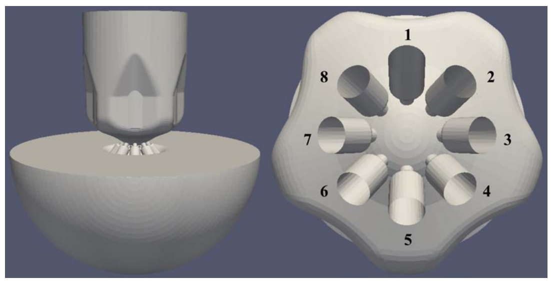


Figure 1: Spray G nozzle geometry with 8 nozzle holes and 5 dimples obtained from ECN [10].

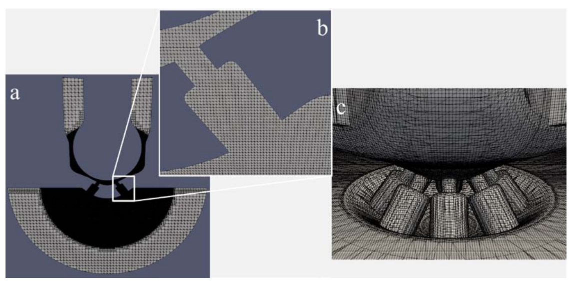


Figure 2: (a) Vertical cut-plane showing the mesh with 9 mm hemispherical outlet domain at fully opened conditions, (b) Zoomed in view of mesh in the nozzle region and (c) 3D mesh in the nozzle region.

Initial and boundary conditions

At the inlet and outlet, a pressure boundary condition was used. For walls and the interface between the liquid and vapor phase, a no-slip boundary condition was enabled. Turbulent kinetic energy and dissipation rate were given as the turbulence boundary condition at both the inlet and outlet of the domain. As liquid fuel cannot be 100% pure, a very small amount of non-condensable gas was considered to be present in the liquid fuel. This non-condensable gas also acts as nucleation sites for cavitation inception. A more realistic needle lift profile obtained from experiments was used in the simulation. The needle motion was changed to make it monotonic for a finite positive initial needle lift. The needle position was started from 2 μm of lift and remained stationary until 2.5 μs into the simulation. The time of 2.5 μs was chosen because this was the amount of time the experimental data indicated it would take to reach 2 μm of lift. After this point, the needle position followed the experimental lift generated by the experiment [10]. Fig. 3 shows the experimental ensembled-averaged needle lift and lift profile used in the simulations. The details of the cases studied in this work are summarized in Table 1.

Table 1: Cases simulated.

	Spray-G	Spray-G2
Fuel	*iso*-octane	*iso*-octane
Fuel temperature	363.15 K	363.15 K
Injection pressure	20 MPa	20 MPa
Back pressure	600 kPa	53 kPa
Ambient temperature	573.15 K	333.15 K

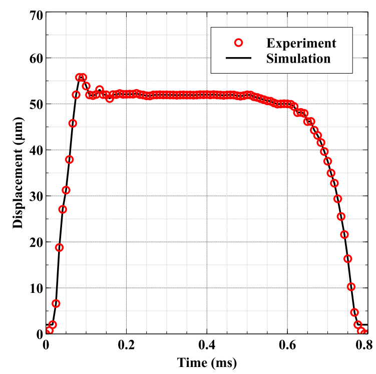


Figure 3: Experimental [10] ensembled-averaged needle lift and needle lift profile used in simulation which starts at 2 μm.

Results and discussion

The numerical simulation results were validated quantitatively and qualitatively against the experimental results available from literature. Fig. 4 shows the qualitative comparison of both flashing and non-flashing conditions. The volume rendered fuel mass fraction of *iso*-octane is compared with the spray images obtained from the experiments. It can be observed that there is a uniform contact of fuel with counter-bore under flashing conditions, while non-flashing case shows gaps between the plumes allowing ingestion of ambient gas. This phenomenon is very well captured by the simulations.

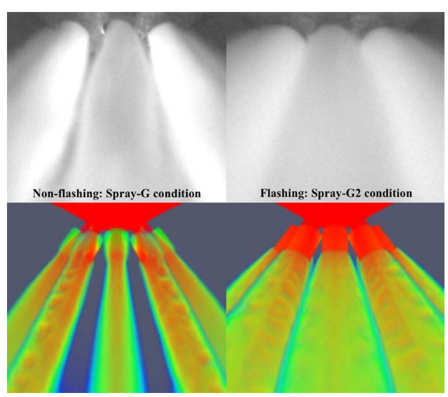
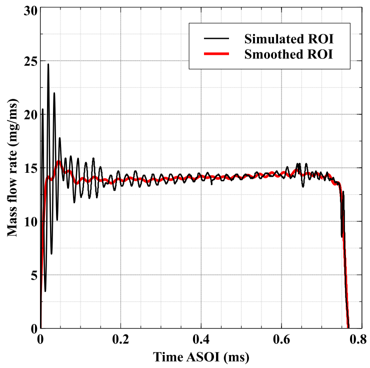
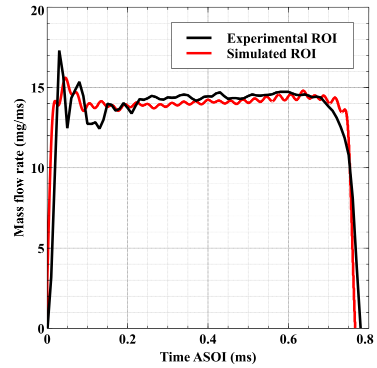


Figure 4: Experimental images (top) [6] of non-flashing (left) and flashing (right) and volume rendered fuel mass fraction from the simulations (bottom).

Fig. 5a shows the raw and smoothed rate of injection profiles obtained from the simulation. The raw data exhibited a high degree of transient variability as an inherent nature of two-phase flow. The raw data have been filtered using a Fast Fourier Transform (FFT) method to de-noise the simulation result. Fig. 5b, on the other hand, shows the comparison between experimental and smoothed simulated data. It can be found that there is a good agreement between the experimental and simulated data. This demonstrates the capability of the methodology to predict the rate of injection profile under flash boiling conditions incorporating transient needle motion.



(a) Raw predicted rate of injection and smoothed rate of injection using FFT filter.



(b) Comparison of experimental and simulated rate of injection.

Figure 5: Rate of injection (ROI).

The total injected mass was calculated by integrating the mass flow rate across the outlet of each hole throughout the injection duration. The total mass agreed well with experimental data of 10 mg as specified by ECN. The hole-to-hole variation is shown in Fig. 6. It was found that the variation was very small in the order of 2-3%. No apparent dependance was found on the asymmetry of the nozzle, as expected in previous studies [4,5]. This indicates that the nozzle asymmetry has negligible importance on hole-to-hole variations and if any were found experimentally, it may be due to the manufacturing imperfections in the nozzle.

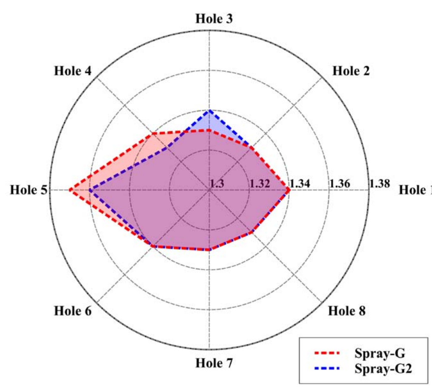


Figure 6: Hole-to-hole variation in mass flow rate.

Fig. 7 shows the comparison of liquid spray half-cone angle between simulation and experiment. As the phase change occurs inside the nozzle holes, it forces liquid to flow out at a different angle and it displayed a good agreement with the experimental result.

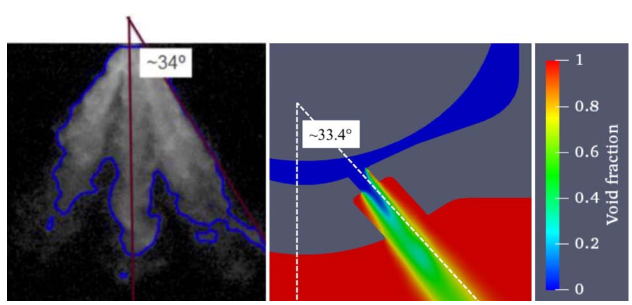


Figure 7: Comparison of liquid spray half cone angle between simulation and experiment [11].

Fig. 8 shows the comparison of cavitation inception between spray-G and spray-G2 conditions. It can be found that mild cavitation like phenomenon takes place inside the injector nozzle holes. The higher differential pressure in the spray-G2 case under same fuel temperature leads to more vapor formation inside the holes as compared to the spray-G case.

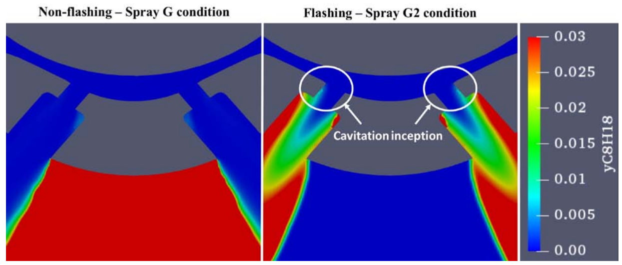


Figure 8: Comparison of cavitation inception between Spray-G and Spray-G2 conditions.

Conclusions

Numerical simulations of internal and near nozzle flow under flashing and non-flashing conditions with transient needle motion were succesfully done with the CONVERGE v2.4 CFD code. The simulations predicted the plume-to-plume interactions under flashing conditions and rate of injection in good agreement with the experiments. It was found that the hole-to-hole variation due to nozzle asymmetry was not significant. Furthermore, the simulations captured mild cavitation like phenomenon inside the nozzle holes due to higher differential pressure.

References

[1] Zhang, Y., Kumar, P., Traver, M., Cleary, D. (2016) *Conventional and Low Temperature Combustion Using Naphtha Fuels in a Multi-Cylinder Heavy-Duty Diesel Engine.* SAE International Journal of Engines. 9.
[2] Neroorkar, K. (2011). *Modeling of flash boiling flows in injectors with gasoline-ethanol fuel blends.* University of Massachusetts Amherst.
[3] Saha, K., Som, S., Battistoni, M., Li, Y., Quan, S. and Senecal, P.K. (2015). *Numerical simulation of internal and near-nozzle flow of a gasoline direct injection fuel injector.* In Journal of Physics: Conference Series. 656(1).
[4] Saha, K., Som, S., Battistoni, M., Li, Y., Quan, S. and Senecal, P.K. (2016). *Modeling of internal and near-nozzle flow for a gasoline direct injection fuel injector.* Journal of Energy Resources Technology. 138(5).
[5] Saha, K., Som, S., Battistoni, M., Li, Y., Pomraning, E. and Senecal, P.K. (2016). *Numerical investigation of two-phase flow evolution of in-and near-nozzle regions of a gasoline direct injection engine during needle transients.* SAE International Journal of Engines. 9(2016-01-0870).
[6] Moulai, M., Grover, R., Parrish, S. and Schmidt, D. (2015). *Internal and near-nozzle flow in a multi-hole gasoline injector under flashing and non-flashing conditions* SAE Technical Paper. (No. 2015-01-0944).
[7] Baldwin, E.T., Grover, R.O., Parrish, S.E., Duke, D.J., Matusik, K.E., Powell, C.F., Kastengren, A.L. and Schmidt, D.P. (2016). *String flash-boiling in gasoline direct injection simulations with transient needle motion.* International Journal of Multiphase Flow. 87.
[8] Schmidt, D., Rakshit, S., and Neroorkar, K. (2009). *Thermal and Inertial Equilibrium in Small, High-Speed, Cavitating Nozzle Simulations.* 11th International Conference on Liquid Atomization and Spray Systems (ICLASS-2009).
[9] Bilicki, Z. and Kestin, J. (1990). *Physical aspects of the relaxation model in two-phase flow.* Proceedings of the Royal Society of London A: Mathematical, Physical and Engineering Sciences. 428.
[10] Engine Combustion Network. (accesed on 29 October, 2017). *https://ecn.sandia.gov/gasoline-spray-combustion/target-condition/spray-g-operating-condition/.*
[11] Manin, J., Jung, Y., Skeen, S., Pickett, L. et al. (2015). *Experimental Characterization of DI Gasoline Injection Processes*. SAE Technical Paper. (No. 2015-01-1894).

Acknowledgements

This work was sponsored by King Abdullah University of Science and Technology. The computational simulations utilized the clusters at KAUST Supercomputing Laboratory. The authors thank Convergent Science Inc. for providing the CONVERGE license.