

Numerical Simulations of Supersonic Diesel Spray Injection and the Induced Shock Waves

Shaoping Quan, Meizhong Dai, Eric Pomraning, P. K. Senecal, and Keith Richards
Convergent Science Inc.

Sibendu Som
Argonne National Laboratory

Scott Skeen, Julien Manin, and Lyle M. Pickett
Sandia National Laboratories

ABSTRACT

Shock waves have been recently observed in high-pressure diesel sprays. In this paper, three-dimensional numerical simulations of supersonic diesel spray injection have been performed to investigate the underlying dynamics of the induced shock waves and their interactions with the spray. A Volume-of-Fluid based method in the CFD software (CONVERGE) is used to model this multiphase phenomena. An adaptive Mesh Refinement (AMR) scheme is employed to capture the front of the spray and the shock waves with high fidelity. Simulation results are compared to the available experimental observations to validate the numerical procedure. Parametric studies with different injection and ambient conditions are conducted to examine the effect of these factors on the generation of shock waves and their dynamics.

CITATION: Quan, S., Dai, M., Pomraning, E., Senecal, P. et al., "Numerical Simulations of Supersonic Diesel Spray Injection and the Induced Shock Waves," *SAE Int. J. Engines* 7(2):2014, doi:10.4271/2014-01-1423.

INTRODUCTION

Shock waves are very common in a number of engineering applications and natural phenomena, such as supersonic flying bullets and man-made flight vehicles, detonation, and meteor entering events, and have been extensively studied for around a century. However, studies on shockwaves in high pressure liquid jets only started at the end of the last century, and the literature on this topic is very limited.

High pressure fuel injection has become popular in modern diesel engines as it is believed that a high fuel injection pressure increases the efficiency of the combustion and thus reduces emissions. The fuel injection pressure can be as high as several hundred mega Pascal (MPa). Under such a high injection pressure, the maximum velocity of the diesel spray can be supersonic, which may generate shock waves. Nakahira et al [1] performed experiments on high pressure fuel injection with a maximum injection pressure of 250 MPa. Shock waves including detached bow shocks were observed around the spray for an injection pressure greater than 80 MPa. Nishida et al. [2] reported shock waves around the spray in their diesel fuel spray injection experiments with the injection pressure up to 300 MPa. Using synchrotron x-radiography and a fast x-ray detector, MacPhee et al. [3] observed attached

shock waves at the tip of the spray in their high-pressure common-rail diesel injection system using an injection pressure of 135 MPa. Im et al. [4] numerically simulated supersonic liquid jets and the induced oblique tip shock waves using a transient multiphase computational fluid dynamics (CFD) code, and their simulations compared well to their experiments. Pianthong et al. [5-6] observed shock waves in supersonic liquid jets and pulsed supersonic fuel jets. The high speed jets were created by a vertical powder gun, and the supersonic jets have a speed up to 2000 m/s. Milton and Pianthong [7] reviewed the characteristics and potential of supersonic, pulsed liquid jets for fuel injection.

Recently, Kook and Pickett [8] investigated shock wave generation by high-pressure diesel sprays over a wide range of ambient temperature and density in a high-temperature, high-pressure optical vessel. Using high-speed schlieren imaging, these authors captured injection-resolved shock waves at an injection pressure of 150 MPa. The authors concluded that both high ambient temperature and density inhibit shock wave generation because: 1) high gas temperature increases the speed of sound as the speed of sound based on the ambient fluid properties is proportional to the square-root of the temperature (for an ideal gas the speed of sound is $c = \sqrt{\gamma RT / M}$, where γ is the ratio of specific heat,

R denotes the universal gas constant, T stands for the gas temperature, and M is the molecular weight of the ambient gas); 2) high gas density decelerates the spray as much momentum is transferred from the injected liquid fuel to the gas to accelerate the gas phase.

It is noted that most of the previous studies were experiments. To the best of our knowledge the only numerical simulation performed by Im *et al.* [4] used a discrete particle scheme in a Lagrangian coordinate system to calculate the liquid jet and an Eulerian coordinate system to compute the gas phase, which is usually called Lagrangian-Eulerian method (LE).

In this paper, we present an Eulerian-Eulerian method (EE) to simulate supersonic liquid injection in an otherwise initially quiescent ambient gas. In this method, both the liquid and gas phases are solved in the same Eulerian coordinate system; and the interface between these two phases is computed using a Volume of Fluid (VOF) based scheme. At such high velocities, the fluid flow must be turbulent, and a dynamic structure Large Eddy Simulation (LES) model [9,10] is employed in the simulations. The objective of this study is to investigate the effects of ambient pressure and the injection rate on the spray dynamics and the induced shock waves using CFD simulations.

EXPERIMENTAL SET-UP

High-speed schlieren imaging in a pressurized spray chamber was performed to visualize the spray development and shock wave formation, as shown in Figure 1. The schlieren technique responds to gradients in refractive index, formed at the shock wave boundary or within the spray. The current experiments closely follow that of Kook and Pickett [8], except with a number of improvements that have been enabled by advances in lighting and camera technology. Imaging was performed at 180,000 frames per second with a resolution of 4.8 pixels/mm, which is more than a factor of two improvement in both speed and resolution. In addition, a high-speed LED producing very short (1 μ s) light pulses was operated synchronously with the camera to nearly freeze motion in each image. As described in [8], the instantaneous schlieren images were normalized by the light intensity present at the time of injection.

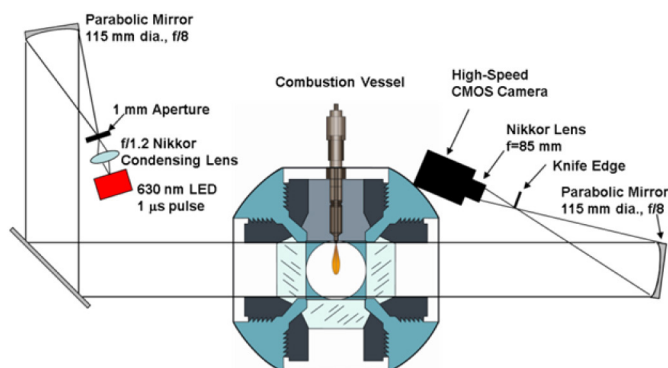


Figure 1. Spray chamber with setup for high-speed schlieren imaging.

The experimental conditions (see Table 1) closely mimic that of Ref. [8] with several exceptions. The fuel injector nozzle is the same, but it is now mounted on an injector body used in research for the Engine Combustion Network where a massive data set documenting the spray development and combustion has been developed (www.sandia.gov/ECN). Research for the ECN requires that fuel properties are well known, therefore, the experiments were performed using n-dodecane fuel, rather than a generic diesel fuel used by Ref. [8]. All other fuel and ambient conditions are matched to that shown by Kook and Pickett [8] to offer the greatest opportunity for synergy and comparison.

A benefit of using ECN standard conditions and n-dodecane fuel is that there is ongoing work to understand the actual rate of injection during very fast injector opening transients [11]. Based on work performed for ECN Spray A injectors applying different diagnostics and modeling techniques and when using n-dodecane, a fuel mass flow rate application was developed and is openly shared at <http://www.cmt.upv.es/ECN03.aspx>. We applied this tool at the corresponding conditions of this study to generate the rate of injection given in Figure 2. The “high” injection rate corresponds to the 150 MPa common rail setting given in Table 1, where supersonic injection velocities are observed. The “low” injection rate is for 26 MPa injection pressure, producing a much slower spray.

Table 1. Experimental conditions.

Common rail injector	ECN Spray A nozzle 210370
Nozzle diameter	90 μ m with KS1.5/0.86
Fuel	n-dodecane
Rail pressure	150 MPa
Temperature of fuel at injector tip	363 K (10 K uncertainty)
Discharge coefficient	0.89 (@ 150 MPa)
Area contraction coefficient	0.98 (@ 150 MPa)
Ambient gas	N ₂
Ambient temperature	455 K
Ambient gas density	0.81, 2.55, 11.7 kg/m ³

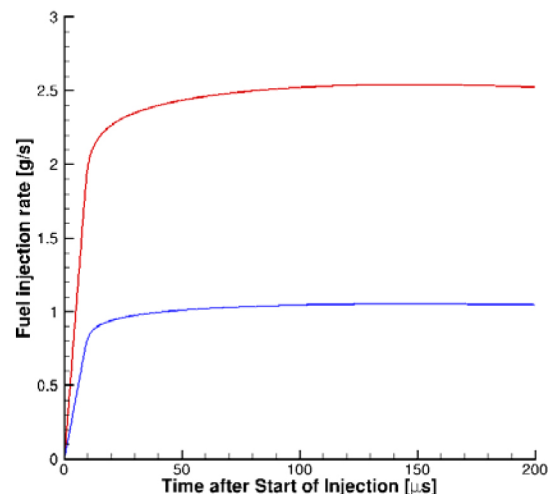


Figure 2. Two injection rates used in the simulations: a high (H) and a low (L) injection rates. The corresponding injection pressures are 150 MPa and 26 MPa, respectively.

GOVERNING EQUATIONS AND NUMERICAL METHODOLOGY

The CONVERGE CFD software package [12], which is a general purpose CFD code for computing three-dimensional, incompressible or compressible, chemically-reacting fluid flows in complex geometries with stationary or moving boundaries, is employed as the computational framework for the simulations. In all the simulations, both the liquid fuel and the ambient gas are assumed to be compressible. This section presents a brief overview of the governing equations and the numerical algorithms.

The governing equations for the EE model recently implemented in CONVERGE involves the conservation of mass, momentum, energy, and species, which are expressed as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = s \quad (1)$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} + s_i \quad (2)$$

$$\begin{aligned} \frac{\partial \rho e}{\partial t} + \frac{\partial \rho u_i e}{\partial x_i} = & -P \frac{\partial u_i}{\partial x_i} + \sigma_{ij} \frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_i} \left(k \frac{\partial T}{\partial x_i} \right) \\ & + \frac{\partial}{\partial x_i} \left(\rho \sum_m D h_m \frac{\partial Y_m}{\partial x_i} \right) + s \end{aligned} \quad (3)$$

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial \rho_m u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\rho D \frac{\partial Y_m}{\partial x_i} \right) + s_m \quad (4)$$

Where ρ is the density of the mixture, ρ_m is the density of species m , σ_{ij} denotes the stress tensor, k stands for the conductivity, Y_m is the mass fraction of species m , D is the mass diffusion coefficient, h_m denotes the species enthalpy, e stands for the specific internal energy, and s is the respective source.

The void fraction, α , is computed using the mass fraction of the liquid and gas as

$$\alpha = \frac{m_g / \rho_g}{m_g / \rho_g + m_l / \rho_l} \quad (5)$$

where the subscripts g and l denote the gas and the liquid respectively, and m here stands for the total mass. It can be concluded that if a cell is filled with gas, then $\alpha=1$; if a cell is

filled with liquid, then $\alpha=0$; otherwise $0<\alpha<1$. Apparently, this void fraction mimics the color function in the Volume of Fluid method. Then we have,

$$\rho = \alpha \rho_g + (1 - \alpha) \rho_l \quad (6)$$

The conservation of mass and momentum equations are solved using a pressure-velocity coupling iteration method, i.e. the Pressure Implicit with Splitting of Operators (PISO) method of Issa [13]. The PISO algorithm starts with a predictor step for an intermediate velocity u^* , and it can be solved using

$$\frac{[\alpha^n \rho_g^n + (1 - \alpha^n) \rho_l^n] (u_i^* - u_i^n)}{dt} = -\frac{\partial P^n}{\partial x_i} + H_i^* \quad (7)$$

where H includes all the convection, diffusion and source terms, the superscripts n and $*$ stand for the previous time-step and the intermediate most up-to-date values, respectively. Please note that both the gas and liquid are assumed compressible. Then an intermediate pressure P^* can be calculated by

$$\begin{aligned} \frac{\partial^2 (P^* - P^n)}{\partial x_i^2} - \frac{[\alpha^* \phi_g^n + (1 - \alpha^*) \phi_l^n] (P^* - P^n)}{dt^2} = \\ \frac{1}{dt} \left(\frac{\partial \rho^n u_i^*}{\partial x_i} - S \right) + \frac{[\alpha^* \phi_g^n + (1 - \alpha^*) \phi_l^n] P^n}{dt^2} - \frac{\rho^n}{dt^2} \end{aligned} \quad (8)$$

where $\rho^n = \alpha^n \rho_g^n + (1 - \alpha^n) \rho_l^n$, ϕ_l and ϕ_g are the pressure ratios for the liquid and the gas, respectively, and can be calculated from the equations of state of the liquid and the gas. The pressure ratios are defined as

$$\begin{aligned} \phi_l &= P / \rho_l \\ \phi_g &= P / \rho_g \end{aligned} \quad (9)$$

In the simulations, the equations of state for the liquid and the gas are

$$\begin{aligned} \rho_l &= \rho_{l,0} e^{\frac{P - P_{l,0}}{K}} \\ \rho_g &= \frac{P}{R_{specific} T} \end{aligned} \quad (10)$$

where $\rho_{l,0}$ and $P_{l,0}$ are the reference density and pressure for the liquid, respectively, K stands for the liquid bulk modulus, and $R_{specific}$ denotes the specific gas constant.

