Collaborative Investigation of the Internal Flow and Near-nozzle Flow of a 8-Hole Gasoline Injector(ECN Spray G)

Journal Title
XX(X):1–15
©The Author(s) 2019
Reprints and permission:
sagepub.co.uk/journalsPermissions.nav
DOI: 10.1177/ToBeAssigned
www.sagepub.com/

SAGE

Chinmoy K. Mohapatra¹, David P. Schmidt¹, Brandon A. Sforzo², Katarzyna E. Matusik², Zongyu Yue², Christopher F. Powell², Sibendu Som², Balaji Mohan³, Hong G. Im³, Jihad Badra⁴, Mathis Bode⁵, Heinz Pitsch⁵, Dimitrios Papoulias⁶, Kshitij Neroorkar⁷, Samir Muzaferija⁸, Pedro Martí-Aldaraví⁹, María Martínez⁹

Abstract

The internal details of fuel injectors have a profound impact on the emissions from gasoline direct injection (GDI) engines. However, the impact of injector design features is not currently understood, due to the difficulty in observing and modeling internal injector flows. GDI flows involve moving geometry, flash-boiling, and high levels of turbulent two-phase mixing. In order to better simulate these injectors, five different modeling approaches have been employed to study the Engine Combustion Network (ECN) Spray G injector. These simulation results have been compared to experimental measurements obtained, among other techniques, with X-ray diagnostics, allowing the predictions to be evaluated and critiqued. The ability of the models to predict mass flow rate through the injector is confirmed, but other features of the predictions vary in their accuracy. The prediction of plume width and fuel mass distribution vary widely, with Volume-of-Fluid (VOF) tending to overly concentrate the fuel. All the simulations, however, seem to struggle with predicting fuel dispersion and by inference, jet velocity. This shortcoming of the predictions suggest a need to improve Eulerian modeling of dense fuel jets.

Keywords

Spray simulation, GDI, Flash boiling, Cavitation, ECN, Nozzle Flow

Introduction

Gasoline Direct Injection (GDI) offers the opportunity to carefully tailor the delivery of fuel. A designer could hypothetically control where and when fuel arrives in the cylinder. However, in order to take advantage of this opportunity, we must have the capability of predicting how a specific injector will deliver fuel and how this fuel will atomize and mix with surrounding air. This predictive ability depends on our understanding, as a community, of the physics governing the GDI process.

The Engine Combustion Network (ECN) has provided the engine community with an opportunity to openly share data describing a common scientific target: the Spray G injector. This eight-hole injector has been widely studied, both experimentally and computationally. The injector provides an open platform on which we may build our scientific understanding.

This paper reviews the contributions made on internal and near-nozzle flow under the standard conditions stipulated by the Spray G target during the 6th meeting of the Engine Combustion Network in Valencia, Spain in September of 2018. The paper validates, and occasionally invalidates, the model predictions. One of the more interesting opportunities afforded to us is the chance to assess the repeatability of modeling technology. If a model is transferred from an academic source to an industrial Computational Fluid Dynamics(CFD) code, can the predictive ability of the model

be reproduced? The results also reveal the limitations of the state of the art.

Model Description

Several institutions contributed results generated by either in-house or commercial CFD codes. The following section reviews the software and methods that generated these results. There are the expected variations in mesh resolutions and meshing strategies. Careful examination also reveal

¹Department of Mechanical and Industrial Engineering, University of Massachusetts, Amherst, USA

²Argonne National Laboratory, Lemont, IL, USA

³Clean Combustion research Center, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia

⁴Fuel Technology Division, R&DC, Saudi Aramco, Dhahran, Saudi Arabia

⁵Institute for Combustion Technology, RWTH Aachen University, Aachen, Germany

⁶Siemens Industries Software Computational Dynamics Ltd., London, UK

⁷Siemens Industries Software Computational Dynamics Ltd., Bangalore, India

⁸Siemens Industries Software GmbH, Nuremberg, Germany

⁹CMT- Motores Térmicos, Universitat Politècnica de Valencia, Valencia, Spain

Corresponding author:

Chinmoy K. Mohapatra, Mechanical and Industrial Engineering University of Massachusetts, Amherst, 160 Governors Dr., Amherst, MA, 01003, USA

Email: cmohapatra@umass.edu

differences in how nominally identical models, such as those used for phase change, vary from code to code. In some cases, multiple institutions used the same CFD code, allowing for variation due to user preferences. In another case, one institution employed two different codes: CMT submitted results from both CONVERGE and Star CCM+. A summary of the contributions is given in Table 1 and explained in detail below. The explanations are organized by the code.

The University of Massachusetts Amherst

An Eulerian approach based on the Homogeneous Relaxation Model developed by Schmidt et al.¹, Gopalakrishnan et al.² and Neroorkar et al.³⁻⁵ has been used for the current study by the University of Massachusetts Amherst (UMass). The in-house solver, known as HRMFOAM, is implemented using the foam-extend branch of the OpenFOAM CFD library. Several past publications^{1,6-10} have successfully validated the HRMFOAM code for many practical applications such as channel flow, condensing two-phase injector flow, cavitating diesel flow, and flash boiling GDI flow.

The governing equations for mass and momentum conservation are described in Equation. (1) and (2) respectively. Both equations are written using barycentric velocity. All transported variables solved by the following formulations are Favre averaged to account for turbulence and variable density.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \phi = 0 \quad (1)$$

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot (\phi \vec{U}) = -\nabla p + \nabla \bar{\tau} + \bar{f} \quad (2)$$

In the above equations, ϕ represents the face valued mass flux. $\bar{\tau}$ represents the stress tensor, which includes both the viscous and turbulent stresses. The volumetric source term, $\bar{f}$, in Equation (2) corresponds to an artificial drag force. This added force is a component of a needle seat-sealing algorithm, which is activated when the needle is nearly closed against the wall. This force is applied in the narrow region between the needle and seat only during periods of very low lift. The details of the sealing algorithm is explained by Mohapatra et al.¹¹.

To account for the mixing of ambient air and fuel, a transport equation for the mean non-condensable gas mass fraction, $\tilde{Y}$, is solved as shown in Equation. (3).

$$\frac{\partial \rho \tilde{Y}}{\partial t} + \nabla \cdot (\phi \tilde{Y}) = \nabla \cdot (\rho \tilde{u'Y'}) \quad (3)$$

In Equation. (3), the turbulent diffusion gas flux term, $\tilde{u'Y'}$, accounts for the mixing effect of the relative velocity between the two phases. This term is closed using Fick's law of diffusion, as seen in Equation. (4).

$$\rho \tilde{u'Y'} = \frac{\mu_t}{Sc_r} \nabla \tilde{Y} \quad (4)$$

Here, μ_t and Sc_r are the turbulent viscosity and realizable Schmidt number fields. This formulation is explained by Desantes et al.¹².

The vapor generation is modeled using the HRM, which was proposed by Bilicki and Kestin¹³ based on

the modification to the homogeneous equilibrium model to incorporate the one-dimensional variability of the vapor generation and condensation rates. This model governs the local rate of change of the dryness fraction, which tends towards its equilibrium value, as shown in Equation. (5).

$$\frac{Dx}{Dt} = \frac{\bar{x} - x}{\Theta} \quad (5)$$

Here, $\bar{x}$ is the equilibrium quality, x is the instantaneous quality and Θ is the time scale. The local enthalpy and pressure are inputs for a lookup table generated by the REFPROP database¹⁴ that governs the equilibrium quality.

Based upon Reocreux's "Moby Dick" experiments, Downar-Zapolski et al.¹⁵ proposed a correlation for the time scale Θ , as seen in Equation. (6).

$$\Theta = \Theta_0 \alpha^a \phi^b \quad (6)$$

In Equation. (6), $\Theta_0 = 3.84 \cdot 10^{-7}$ s, α is the fuel void fraction, $a = -0.54$, $b = -1.76$, and ϕ is a dimensionless pressure defined in Equation. (7),

$$\phi = \left| \frac{p_{sat} - p}{p_c - p_{sat}} \right| \quad (7)$$

where p_{sat} is the saturation pressure and p_c is the critical pressure of the fluid.

To allow for the pressure to respond to both compressibility, density change from phase change, and density change from turbulent mixing with the non-condensable gas; HRM is extended to three dimension by connecting predicted phase change with the conservation of mass and momentum through a simple chain rule, as shown in Equation 8.

$$\begin{aligned} \frac{D\rho}{Dt} = & \frac{\partial \rho}{\partial p} \bigg|_{x,h,y} \frac{Dp}{Dt} + \frac{\partial \rho}{\partial x} \bigg|_{p,h,y} \frac{Dx}{Dt} \\ & + \frac{\partial \rho}{\partial h} \bigg|_{p,x,y} \frac{Dh}{Dt} + \frac{\partial \rho}{\partial y} \bigg|_{p,h,x} \frac{Dy}{Dt} \end{aligned} \quad (8)$$

To model turbulence, the SST $k - \omega$ Reynolds-Averaged Navier-Stokes (RANS) model is used by UMass, with log-layer wall functions at the injector walls. This model is well known for resolving separating flows with adverse pressure gradients³⁸. UMass accounts for the compressibility effect and assumes the simulation to be isenthalpic, hence does not solve the energy equation.

CMT and KAUST-Aramco

Both the institutions CMT and KAUST-Aramco performed their internal flow simulations using the CONVERGE v2.4 CFD package. CONVERGE solves the classical conservation equations for mass (Equation 9), momentum (Equation 10), and energy (Equation 11) using a single-fluid approach. In these equations: ρ is the density, $\vec{v}$ is the velocity, p is the pressure, e is the internal energy, $\bar{\tau}$ is the mixture stress-strain tensor, K is the molecular and turbulent conductivity, D_i is the molecular and turbulent diffusion coefficient, h_i is the species enthalpy and Y_i is the mass fraction of each specie.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (9)$$

Table 1. Modeling contributions

Institution	UMass	Siemens	CMT	CMT	Argonne	KAUST-Aramco	RWTH Aachen
Code	HRMFoam	Star CCM+	CONVERGE	Star CCM+	CONVERGE	CONVERGE	in-house code
Cases	G, G2	G, G2	G, G2	G	G	G	G
Compressibility	Yes	No	Yes	No	No	Yes	Yes
Cavitation model	Yes	Yes	Yes	Yes	No	Yes	Yes
Phase change model	HRM	HRM	HRM	HRM	No	HRM	HEM
Turbulence model	k- ω SST	k- ω SST	k- ϵ RNG	k- ω SST	LES dynamic structure	k- ϵ RNG	LES/DNS Dynamic Smagorinsky
Spatial discretization	2 nd order	2 nd order	1 st order	1 st order	-	-	-
Liquid/Gas Interface Model	Eulerian Mixture fluid	Eulerian Mixture fluid	Eulerian Mixture fluid	Eulerian Mixture fluid	VOF PLIC	Eulerian/ Mixture fluid	Eulerian Lagrangian
Heat transfer	Isenthalpic	Adiabatic	Isothermal	Isothermal	Adiabatic	Isothermal	-
Adaptive/static	Static	Static	AMR	Static	AMR	AMR	-
Needle motion	Yes	No	Yes	No	Yes	Yes	Yes

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot \bar{\tau} + \rho\vec{v} + \vec{F}_{sf} \quad (10)$$

$$\begin{aligned} \frac{(\rho e)}{\partial t} + \nabla \cdot (\rho\vec{v}e) &= -p\nabla \cdot \vec{v} + \bar{\tau} \cdot \nabla \vec{v} + \\ &+ \nabla \cdot (K\nabla T) + \nabla \cdot \left(\rho D \sum_i h_i \nabla Y_i \right) \end{aligned} \quad (11)$$

Species are solved in first instance through the species transport Equation 12, and then the void fraction is calculated (it is not transported directly) from Equation 13, where subscript 'g' represents the sum of gaseous phases. The new variable which appears in this equation is S_i , the source term related with the mass transfer as a result of phase change.

$$\frac{\partial(\rho Y_i)}{\partial t} + \nabla \cdot (\rho\vec{v}Y_i) = \nabla \cdot (\rho D_i \nabla Y_i) + S_i \quad (12)$$

$$\alpha_g = \frac{\left(\frac{Y_g}{\rho_g} \right)}{\sum \frac{Y_i}{\rho_i}} \quad (13)$$

Three different species are considered in the current work: the liquid fuel phase (number 1), the vapor fuel phase (number 2), and the ambient non-condensable gas phase (number 3). The density of the mixture, as well as other properties, is computed as shown in Equation 14 through the volume fraction α_i and density ρ_i of each of the three species.

$$\rho = \alpha_1 \rho_1 + \alpha_2 \rho_2 + \alpha_3 \rho_3 = \alpha_g \rho_g + (1 - \alpha_g) \rho_1 \quad (14)$$

A Volume-of-Fluid (VOF) interface-capturing method, such as it is described by Battistoni et al.¹⁶, is used to simulate the multiphase flow inside and outside the nozzle. In order to compute the void fraction of each cell, a function α is used, so that, $\alpha = 0$ corresponds to those cells which are filled with only liquid, $\alpha = 1$ to cells with only gas and α between 0 and 1 refers to cells filled with a mix of liquid and gaseous species.

In order to obtain the mass transfer term S_v from liquid to vapor fuel, a slightly modified form of the non-equilibrium HRM of Bilicki and Kestin¹³ and Schmidt¹ is used. This model is selected because it has been proved that this

approach is able to compute phase change due to not only cavitation but also flash-boiling. Summarizing, for the HRM, the rate of change of local vapor quality (x), as given in Equation 15, provides the estimate of S_v .

$$\frac{Dx}{Dt} = F \frac{\bar{x} - x}{\theta} \quad (15)$$

In this equation, $\bar{x}$ represents the equilibrium quality, determined by Equation 16, where h_l is the saturated liquid enthalpy and h_v is the saturated vapor enthalpy, both at the local pressure value. Unlike the other implementations, the CONVERGE implementation adds the parameter F which is set to unity for evaporation and a value of 5000 for condensation. The effect of switching F is to make condensation far slower than evaporation.

$$\bar{x} = \frac{h - h_l}{h_v - h_l} \quad (16)$$

The variable θ is a phase change time scale, obtained with Equation 17, where ψ is given by Equation 18 (being p_{crit} and p_{sat} the critical and saturation pressures, correspondingly), and, according to¹⁷ $\theta_0 = 3.87 \cdot 10^{-7} S$, $a = -0.54$ and $b = 1.76$. One difference between the CONVERGE implementation and the original used in Baldwin et al.⁶ is the definition of α . Here, α is the volume fraction of all gas species while in the original implementation, α was defined as the volume of fuel vapor divided by the volume of fuel.

$$\theta = \theta_0 \alpha^a \psi^b \quad (17)$$

$$\psi = \left| \frac{p_{sat} - p}{p_{crit} - p_{sat}} \right| \quad (18)$$

Finally, in order to take into account the turbulence in the simulations, a Reynolds Averaged Navier-Stokes (RANS) approach is employed by CMT. It requires a closure model in order to compute the turbulent strain-stress tensor. From all the choices available in the literature, the Renormalization Group (RNG) k - ϵ sub-model of Yakhot and Orszag¹⁸ is applied for computing the turbulent kinetic energy k and the turbulent dissipation rate ϵ when CONVERGE software is

used; whilst the Shear-Stress Transport (SST) $k-\omega$ model of Menter¹⁹ is used in the case of StarCCM+. KAUST-Aramco also used the RANS approach, in particular, the Standard $k-\varepsilon$ model for the turbulence closure.

Siemens-CD Adapco and CMT

An incompressible HRM solver implemented in Star CCM+ is used by Siemens-CD Adapco and CMT. The details of their governing equations are mentioned below, which are similar to the UMass implementation of HRMFoam.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (19)$$

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \otimes \vec{v} - \bar{\tau}) = \rho g \quad (20)$$

In the Equation 19 and 20 ρ is the mixture density and $\bar{\tau}$ is the stress tensor including the pressure term. The thermodynamic properties of the mixture are volume-averaged in terms of the local volume-fraction α as shown in Equation 21, where the subscript i denotes the different flow phase in the mixture. By definition, the sum of volume-fractions in each cell is always equal to one. The phase mass conservation equation is expressed in the Equation 22.

$$\rho = \sum_i \alpha_i \rho_i \quad (21)$$

$$\frac{\partial (\rho_i \alpha_i)}{\partial t} + \nabla \cdot \rho_i (\alpha_i \vec{v}) = S_{\alpha_i} \quad (22)$$

The extra term S_{α_i} added to the right hand side of the Equation 22 represents the source/sink term for phase i , due to flash-boiling effects. Siemens use the HRM similar to the UMass implementation for the phase-change process. To account for the turbulence closure Siemens use the two-equation Shear-Stress Transport (SST) model.

Argonne

VOF method²⁰ is applied to model the two-phase flow in the study by Argonne using the CONVERGE code²¹. In this method, both fluids are resolved on computational mesh and velocity difference across inter-phase is neglected. The governing equation reads,

$$\rho \left[\frac{\partial u}{\partial t} + u \cdot \nabla u \right] = -\nabla p + \nabla \cdot [\mu \nabla u + \mu (\nabla u)^T] + \sigma \kappa \delta_s n \quad (23)$$

Where, u is the velocity field, ρ is the density, p is the pressure, μ is the viscosity, σ is the surface tension, κ is the local interface curvature, δ_s is a Dirac distribution concentrated at interface s , and n is the unit vector normal to the interface.

In a cell where both phases exist, the following equations hold,

$$\rho = \rho_g \alpha + \rho_l (1 - \alpha) \quad (24)$$

$$\mu = \mu_g \alpha + \mu_l (1 - \alpha) \quad (25)$$

The subscripts g and l represent gas phase and liquid phase, respectively. α is the void fraction in a cell. In VOF

method, the transport equation of α is solved for two-phase immiscible incompressible flows.

$$\frac{\partial \alpha}{\partial t} + u \cdot \nabla \alpha = 0 \quad (26)$$

Additionally, the Piecewise-Linear Interface Calculation (PLIC) method²² is applied to construct sharp interface between phases.

Large-Eddy Simulation (LES) method is used for the turbulent flow modeling. After applying the spatial filtering operation, the Navier-Stokes equation contains an unclosed non-linear term that must be modeled. Dynamic structure model²³ is employed which solves the transport equation for sub-grid scale kinetic energy, k_{sgs} , and the subgrid stress is modeled as:

$$\tau_{ij} = C_{ij} k_{sgs} \quad (27)$$

$$C_{ij} = 2 \frac{L_{ij}}{L_{kk}} \quad (28)$$

Where, L_{ij} is the modified Leonard stress term.

RWTH Aachen

Simulations by RWTH Aachen were conducted using the in-house code CIAO and a combined Large-Eddy Simulation/Direct Numerical Simulation (LES/DNS) as well as Euler/Lagrangian approach. The nozzle internal flow simulations rely on a hybrid solver²⁴, which solves the Navier-Stokes equations in compressible form on a staggered, Cartesian mesh. Phase change is modeled by a tabulated equation-of-state (EOS) approach assuming homogeneous equilibrium²⁵ and solving an additional transport equation for the liquid mass fraction using a WENO5 scheme. Given a temperature and pressure range $\Omega_{T,p} = [T_{\min}, T_{\max}] \times [p_{\min}, p_{\max}]$, the sets Ω_{ξ^v} and Ω_{ξ^s} in dual space are chosen such that

$$n_{T,\text{rel}} = \{|\Omega_{\xi^v}^T|/|\Omega_{\xi^v}| \mid T_{\min} \leq \xi_i^v \leq T_{\max}, \forall \xi_i^v \in \Omega_{\xi^v}^T, \Omega_{\xi^v}^T \subseteq \Omega_{\xi^v}\} \quad (29)$$

and

$$n_{p,\text{rel}} = \{|\Omega_{\xi^s}^p|/|\Omega_{\xi^s}| \mid -p_{\max} \leq \xi_i^s \leq -p_{\min}, \forall \xi_i^s \in \Omega_{\xi^s}^p, \Omega_{\xi^s}^p \subseteq \Omega_{\xi^s}\} \quad (30)$$

are greater than some prescribed values and $\xi_1^v \leq \min_j \xi_1^{v,j}$, $\xi_{N_{\xi^v}}^v \geq \max_j \xi_{N_{\xi^v}}^{v,j}$ and $\xi_1^s \leq \min_i \xi_1^{s,i}$, $\xi_{N_{\xi^s}}^s \geq \max_i \xi_{N_{\xi^s}}^{s,i}$ with n as number of points for the specified dimension for the tabulation. The superscripts j and i indicate their belonging to the sets $\Omega_{\xi^v}^j$ and $\Omega_{\xi^v}^i$ and the latter condition ensures that the domain of the bi-conjugate of the specific internal energy e^{**} matches with that of e . In order to consider sub-grid turbulence contributions, the nozzle internal flow is computed as LES with dynamic Smagorinsky model and coupled to DNS of the primary breakup^{26,27}. Due to the Cartesian mesh and resolution requirements in space and time for the primary breakup DNS, hybrid-parallelization²⁸ as well as work load distribution on node level are used²⁹ in order to reduce the time-to-solution on current supercomputers.

ECN Spray G geometry details and operating condition

The ECN Spray G nozzle consists of a multi-hole nozzle with 8 counter-bore orifices. Its geometric details are listed in Table 2. For these simulations carried out in HRMfoam, CONVERGE, and Star CCM+, the specified nominal values are used to build the computational domain (named Generation 1 geometry), shown in Figure 1. The spherical shape of the needle is also built from the nominal manufacturing dimensions. A semi-spherical volume is added as discharge plenum in order to simulate the first millimeters of the spray development. UMass, KAUST-Aramco and Siemens used a 9mm plenum diameter, however CMT used 6mm plenum diameter for the CONVERGE simulations and 9 mm plenum diameter for the Star CCM+ calculations.

Table 2. ECN Spray G nominal nozzle geometry parameters.

Nozzle type	valve covered orifice (VCO)
Bend angle	0°
Number of holes	8
Orifice shape	circular
Hole shape	straight
Nozzle shape	step hole
L/D ratio	1.4
Orifice diameter	0.165 mm
Orifice length	0.16 – 0.18 mm
Step diameter	0.388 mm
Orifice drill angle	37° relative to nozzle axis
Full outer spray angle	80°

However, experimentalists³⁰ have observed variations in the intricate geometrical details of different Spray G injectors. Argonne in their sector simulation using CONVERGE have incorporated X-ray measured hole 5 geometrical details leading to an expensive computation.

Figure 1. ECN Spray G computational domain with 6 mm semi-spherical discharge plenum. It shows the distribution of the 8-holes and the 5 dimples. The colors represent the different boundary conditions.

Table 3 lists the operating condition for both standard non flashing and flash boiling condition. Both conditions have the same injection pressure and fuel temperature, but the ambient density and back pressure are different. For the G2 condition the back pressure is below the fuel saturation pressure leading to flash boiling conditions. Figure 2 includes the profile for the needle displacement from the initial position measured by X-ray technique. This profile is used for the moving mesh simulations, while

Table 3. Operating condition for G (Non-Flashing) and G2 (Flashing) condition.

Parameter	G-Standard	G2-Flash boiling
Fuel	Iso-Octane	Iso-Octane
Fuel temperature	90 °C	90 °C
Ambient temperature	573 K	333 K
Fuel pressure	20 MPa	20 MPa
Ambient density	3.5 kg m ⁻³	0.5 kg m ⁻³
Ambient Pressure	600 kPa.	50 kPa

accounting for the different initial mesh lift by different institutions.

Figure 2. Ensemble-averaged needle lift (z-axis) profile from the X-ray

Simulation setup

UMass

A primarily hexahedral mesh for the nominal geometry with a cell count of 1.44 million has been created using the GridPro meshing tool. The grid spacing is roughly 7 μm in the sac region, as opposed to the spacing 10 μm in the nozzle region. To avoid higher cell counts, an anisotropic refinement in the narrow region between needle and nozzle was performed. The mesh in the seat and nozzle region can be seen in the Figure 3. The mesh is created at an initial needle displacement of 5 μm.

In this study, the Laplacian mesh motion library of FOAM extend is used to lift the needle according to the experimentally measured needle displacement curve, as shown in Figure 2, while accounting for the initial mesh lift. The needle closure model discussed by Mohapatra et al.¹¹ is also used, which gets activated whenever the needle is displaced less than 1 μm from its initial position. Hence, the sealing algorithm is effectively engaged at 6 μm.

Zero-gradient boundary conditions are used for velocity at both the inlet and outlet. For the pressure at the inlet, a total pressure boundary condition is chosen. At the outlet, a transonic total pressure boundary condition is applied. This boundary condition switches between zero-gradient and total-pressure boundary conditions based upon the Mach number at the exit. In the initial condition, the needle

Figure 3. Cut plane of CFD mesh by UMASS showing anisotropic refinement in the narrow region between the needle and the injector wall

is closed and the fluid is stationary, so all pressure drop should occur at the needle seat. To avoid the formation of spurious pressure waves in the flow domain during the start of simulation, a hyperbolic tangent drop of pressure across the needle seat area is applied. The interface between the fuel and the non-condensable gas is placed at the lower bound of the sealing region. This leaves the sac initially filled with non-condensable gas.

CMT

In this institution, CONVERGE is used for the moving mesh simulation, whereas Star CCM+ is used for the fixed mesh simulation. CONVERGE uses an octree method which generates a volume mesh using a top-down approach. This is done automatically at run-time. The cell size inside the nozzle and orifices is $35\text{ }\mu\text{m}$, while in the discharge volume it is $140\text{ }\mu\text{m}$. Next to the nozzle walls, the cell size is reduced to $17.5\text{ }\mu\text{m}$ to better capture the development of the boundary layer. Figure 4 shows the resultant mesh, with a total cell count of 1.02 million cells. Additionally, an adaptive mesh refinement (AMR) strategy is used in the discharge volume and holes regions to maximize both accuracy and computational efficiency. The AMR automatically adjusts the grid at each time-step, adding cells in areas with large gradients and removing cells that are not needed. The velocity is the selected variable to trigger the AMR, with a sub-grid criterion of 1 m/s , and a minimum cell size set to $17.5\text{ }\mu\text{m}$.

Two different mesh strategies are used and tested in Star CCM+. The first one consists of a structured hexahedral mesh. The base size is $140\text{ }\mu\text{m}$, but the grid is refined in areas where velocity, pressure and mass fraction gradients are expected to be large. Figure 5 depicts this first mesh, which contains a total of 11.44 million cells. In the sac, the orifices and next to nozzle walls the cell size is reduced up to $8.75\text{ }\mu\text{m}$. Also in this region, a 3 layers prism layer of a total thickness of $8.75\text{ }\mu\text{m}$ is used for the walls. In the discharge volume, a cone around each spray axis is refined with a cell size of $17.5\text{ }\mu\text{m}$. A broader cone around this one is refined to $35\text{ }\mu\text{m}$.

The second mesh strategy employed in Star CCM+ is a polyhedral mesh, shown in Figure 6. Its main advantage is the reduction of the total cell count, 5.08 millions in this case.

Figure 4. Sketch of the ECN Spray G computational mesh used in CONVERGE used by CMT. It shows only 3 mm of the semi-spherical discharge plenum.

Figure 5. Sketch of the ECN Spray G hexahedral computational mesh used in Star CCM+ by CMT.

The base size is $60\text{ }\mu\text{m}$, but as in the previous one, the grid is refined where it is needed: inside the nozzle and the orifices the selected size is $18\text{ }\mu\text{m}$. The same prism layer as before is applied. In the discharge volume, the cell size increases from the orifices to the outlet surface with a growth rate of 1.05.

Figure 6. Sketch of the ECN Spray G polyhedral computational mesh used in Star CCM+ by CMT.

The final results are the ones mentioned earlier, which are summarized in Table 4.

A constant pressure boundary condition is used for the inlet in CONVERGE. Stagnation inlet is the equivalent type in Star CCM+, whereas pressure outlet boundary condition

Table 4. Mesh information for each of the simulations.

Code	CONVERGE	Star CCM+	Star CCM+
Element type	Hexahedral	Hexahedral	Polyhedral
Min. size nozzle	17.5 μm	8.75 μm	18 μm
Min. size plenum	17.5 μm	17.5 μm	18 μm
Max. size	140 μm	140 μm	60 μm
Cell count	1.02 mill.	11.4 mill.	5.1 mill.
Peak cell count	1.5 mill.	11.4 mill.	5.1 mill.

type is selected for the outlet. Specified values (pressure, turbulent intensity, turbulent length scale...) depend on the injection conditions being simulated. For the stationary cases, the needle is fixed at 45 mm of lift. On the other hand, for transient simulations, the needle movement is imposed and the average values of lift and wobble obtained are used. Parts or regions of the computational domain are not disconnected during the simulation, thus a minimum needle lift value is needed at “closed” position. In this case, a value of 2 μm is selected.

For all, steady and transient, simulations the initial conditions are similar: stagnated flow with constant turbulence parameters. The complete nozzle (holes included) are filled with liquid fuel (iso-octane) at constant injection pressure and temperature. The outlet plenum is filled by non-condensable gas (Nitrogen) at constant discharge pressure and temperature.

KAUST-Aramco

The computational mesh generated for this study using CONVERGE is shown in Figure 7. A base cell size 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 18.75 μm .

Figure 7. (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 - Mesh created in CONVERGE by KAUST-Aramco

At the inlet and outlet, a pressure boundary condition is used. For walls and the interface between the liquid and vapor phase, a non-slip boundary condition is enabled. Turbulent kinetic energy and dissipation rate are 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 is considered to be present in the liquid fuel. This non-condensable gas also acts as nucleation sites for cavitation inception. The needle position is started from 2 μm of lift and remains stationary until 2.5 μs into the simulation. The time of 2.5 μs is chosen

because this was the amount of time the experimental data indicates it would take to reach 2 μm of lift.

Siemens

For the calculations considered in this study, the needle is assumed to be fixed at its nominal maximum lift position i.e. 50 μm . The finite-volume grid used for the spatial discretization of the solution space is illustrated in Figure 8. Overall, the mesh incorporates close to 8 million mixed hexahedral and wall-prism cells, while inside the nozzle holes, sac-volume and across the vicinity of the spray jets the grid is significantly refined, which varies from 5 μm -25 μm .

Figure 8. Grid architecture for the cut cell mesh created in Star CCM+ by Siemens-CD Adapco

In this study, a stagnation inlet and a Dirichlet pressure boundary at the outlet combined with a zero gradient velocity are applied. The fuel liquid and its vapor phase due to boiling are modeled as single-component, incompressible fluids, corresponding to iso-octane. Accordingly, the thermodynamic properties of the modeled materials are taken from the National Institute of Standards and Technology (NIST) database, as a function of the injection conditions. The density variations of the air component in the plenum are modeled using the ideal gas law.

The convection term in the volume-fraction Equation 22 is integrated using two different discretization schemes, including a first-order upwind method as well as the High Resolution Interface Capturing (HRIC) scheme. Discretization in time is facilitated with a second-order backward Euler scheme using time-steps between 0.1 μs and 0.01 μs , depending on the applied spatial resolution. The upstream solution field is initialized from the inlet conditions, assuming that the pressurized fuel occupies the entire needle-seat passage up to the injection holes. Likewise, the initial conditions for the spray domain, which is filled with air, are set according to the environmental outlet boundary. To eliminate the noise of the initial guess as well as to ensure that flow has reached a statistically steady-state, the solution is iterated for sufficient steps starting from a fully-developed single-phase calculation.

Argonne

The GDI nozzle simulated is ECN Spray G injector 28. A full nozzle geometry model with 1.7 μm resolution is measured by X-ray CT imaging at the Advanced Photon Source (APS) of Argonne National Laboratory. A 135° sector domain is used to reduce the computational expense, which includes

three holes and a spherical chamber region with a radius of 4.5 mm, as shown in Figure 9(a). However, due to the memory requirements of the computational solver for the real geometry details, only the parts of sac bottom, hole and counter-bore surfaces of hole 5 are used in simulation as indicated by the red component in Figure 9(b), in conjunction with nominal geometry for the rest parts. Thus, only the spray emitting from the middle hole is used for analysis and discussion, while the two sprays on the sides serve to provide a realistic boundary condition in terms of air-entrainment etc. for the middle one. An enlarged snapshot for hole 5 can be found in Figure 9(c). Other than the nozzle dimensions that deviate from the nominal values as shown in Figure 9 the real geometry captures the surface roughness, which is approximated at order of $5\text{ }\mu\text{m}$. Additionally, manufacturing defects such as the protrusion at hole exit, and void on the counter-bore surface, are also preserved and highlighted by the red circle in Figure 9(c). These details impacts the spray morphology considerably as shown by Yue et al³¹.

Figure 9. Geometry model used for CFD simulation by Argonne

It should be noted that this institution was the first of its kind, which used X-ray scanned nozzle and counterbore geometry for a LES simulation with transient needle motion. In order to avoid any phase change process such as cavitation or flash-boiling, a colder condition with 298 K as ambient temperature and 315 kPa as ambient pressure was used.

A moving needle is considered with the measured profile as shown in Figure 10. The gray box highlights the 0.1 ms period that is simulated, which covers the whole transient period of start-of-injection. The minimum needle lift is set to be $5\text{ }\mu\text{m}$ at the start of simulation. Above this minimum gap, the in-nozzle region is initialized with fuel at 298 K, 19 MPa, with Dirichlet inlet boundary condition. Below the minimum gap, the sac region is initialized with fuel at 298 K, 315 kPa. The regions of hole, counter-bore, and chamber are initialized with nitrogen at 298 K, 315 kPa, with open boundary at chamber outlet. A $2.5\text{ }\mu\text{m}$ boundary embedding is applied to the real geometry part to preserve the realistic surface details, and region embedding with minimum cell size of $5\text{ }\mu\text{m}$ is applied for the internal and near nozzle region. AMR of $5\text{ }\mu\text{m}$ is also enabled in the chamber based on the velocity field. The total cell count is close to 9 million at the beginning, then increases to nearly 35 million as the needle

lifts and liquid fuel is injected into chamber. With this setup, the current 0.1 ms simulation takes nearly 78,000 core hour to finish.

Figure 10. Needle lift profile vs time obtained from experimental measurements used by Argonne

RWTH Aachen

The simulation at RWTH Aachen used a coupled simulation approach comprising of an LES of the full ECN Spray G injector and a DNS of the primary breakup of one hole. While the DNS used box domain with uniform, structured mesh with $2.0\text{ }\mu\text{m}$, the mesh of the LES was refined around the target hole. In order to estimate the coarsest resolution requirement, a mesh sensitivity analysis was employed using the mass flow rate per hole as target quantity as too coarse resolution in some holes often leads to very asymmetric mass flow rates. It was found that a minimum cell size of $14.0\text{ }\mu\text{m}$ is necessary to achieve realistic mass flow rates per hole with the used Cartesian, structured mesh. The target hole was resolved with refined cells of $2.0\text{ }\mu\text{m}$. Pressure boundary conditions were used together with mesh blanking for the needle opening.

Experimental setup

The diagnostic layout for X-ray radiography measurements collected at the 7-BM beamline is illustrated in Figure 11. The application of the radiography methodology to fuel sprays is thoroughly described in previous works^{34,35}. Specifically here, the synchrotron-produced X-ray light was conditioned to $8.0 \pm 0.01\text{ keV}$ mean photon energy ($4\% \Delta E/E$) using a double-multilayer monochromator. The incoming beam was sized to a $1.3 \times 1.3\text{ mm}$ square cross-section with beam-defining slits and the incident beam intensity, I_0 , was measured using a diamond monitor. The beam was then focused using a pair of Kirkpatrick-Baez³⁶ mirrors to a waist of $4 \times 6\text{ }\mu\text{m}$ (V $\times$ H) located at the tip of the injector at the experimental origin. After transmission through the spray region of interest, the beam intensity, I , was measured with a PIN diode. This intensity was measured in synchronization with the repetitive injection timing and time averaged over the storage-ring orbit period of $3.68\text{ }\mu\text{s}$, defining the temporal resolution of the measurement. At each spatial location, this time history was recorded and ensemble averaged for 32 injection events to improve the signal-to-noise.

Figure 11. Schematics of X-ray radiography experimental setup by Argonne

Following the Beer-Lambert law, the projected mass of the spray (M) within the X-ray path was computed from the beam intensity measurements as described by Equation 31.

$$M(x, y, \theta, t) = -\frac{1}{\mu} \log \left[\frac{I(x, y, \theta, t)}{I_0(t)} \right], \quad (31)$$

where μ is the mass attenuation coefficient, which was empirically collected. The parameters $I_0(t)$ and $I(x, y, \theta, t)$ are the measured beam intensities of the incident beam and the transmitted beam, respectively. These measurements were collected for a sequence of locations, transverse to the injector axis, at a fixed distance. Furthermore, transverse scans were collected at a set of rotation angles of the injector to capture projected mass profiles at multiple viewing angles.

Characteristics flow parameters

The dimensionless coefficients defined by Payri et al.³⁹ that describe the behavior or characteristics of the flow inside the injector are as follows:

- **Discharge coefficient.** This coefficient is defined as the real mass flux with regard to the maximum theoretical mass flux. The maximum mass flux is evaluated considering uniform velocity and equal to Bernoulli velocity and using all the geometric section area.

$$C_d = \frac{\dot{m}}{\dot{m}_{th}} = \frac{\dot{m}}{A_o \rho_f u_{theoretical}} \quad (32)$$

- **Momentum coefficient.** In the same way, the momentum coefficient is defined as the real measured momentum flux with regard to the maximum theoretical momentum flux. Where, as before, the maximum momentum flux is evaluated considering the Bernoulli velocity and the geometric area.

$$C_M = \frac{\dot{M}}{\dot{M}_{th}} = \frac{\dot{M}}{A_o \rho_f u_{theoretical}^2} = \frac{\bar{\dot{M}}}{2A_o \Delta p} \quad (33)$$

- **Velocity coefficient.** This coefficient relates the effective velocity to the maximum theoretical Bernoulli velocity. It is calculated with the following equation,

$$C_v = \frac{u_{ef}}{u_{theoretical}} = \frac{u_{ef}}{\sqrt{2\Delta p / \rho_f}} \quad (34)$$

- **Area coefficient.** This coefficient is used for evaluating the reduction of the effective area with regard to geometric one, and it is calculated as

$$C_a = \frac{A_{ef}}{A_o} \quad (35)$$

Relations between coefficients

If in Equation 32, it is replaced the value of the mass flux in function of the effective parameters, Equation 36 is

$$C_d = \frac{A_{ef} \rho_f u_{ef}}{A_o \rho_f u_{theoretical}} \quad (36)$$

Now, combining the Equation 36 with Equation 34 and Equation 35,

$$C_d = C_a C_v \quad (37)$$

Operating in the same way with the momentum coefficient it is obtained

$$C_M = C_a C_v^2 \quad (38)$$

Finally, combining this two relations it is obtained

$$C_v = \frac{C_M}{C_d} \quad (39)$$

Results and discussion

Rate of injection

For validation, rate of injection (ROI) prediction from the CFD models by all the institutions for the standard G condition are compared with the measured ROI provided by ECN⁴⁰ as shown in the Figure 12. The curve for the ROI prediction using Star CCM+ by CMT and Siemens are flat, as those simulations are carried out for a fixed maximum needle lift condition. As seen in the Figure 12, the moving mesh simulations conducted by CMT and KAUST-Aramco using CONVERGE, UMass using OpenFoam, and RWTH Aachen using the in house built code and the static mesh simulation done by Siemens-CD Adapco using Star CCM+ predicts similar ROI compared to the experimental measured ROI. There is a small difference in ROI prediction by Star CCM+, when run by two different institutions i.e. CMT and Siemens CD-Adapco. CMT predicts a higher ROI compared to the experimental measurement, whereas Siemens CD-Adapco does not. This difference in observation can be attributed to the difference in the solver settings, the spatial discretization and the different meshing strategies and mesh sizes. The ROI predicted by UMass is observed to have a smaller injection duration compared to the experimentally observed injection duration, as UMass is the only institution, which modeled the needle closure event and assumed the needle to be in a closed position, whenever the gap at the seat region is lesser than 6 μm .

In order to analyze the steady state operation of the injectors and continue validating the simulations, averaged values of rate of injection when the needle is fully opened are used to compute the discharge coefficient as defined in the previous Section. Table 5 represents not only the averaged mass flow rate and the discharge coefficient but also the existing differences between the experimental and computational data. According to the previous analysis, the greatest deviation is observed in the discharge coefficients

Table 5. Comparison between experimental and computational results in the steady state period (from 0.4 to 0.6 ms ASOI) for non-flashing condition.

	ROI (g/s)	C_d	C_d deviation
Experimental	13.82	0.52	-
CMT-CONVERGE	14.18	0.53	1.9 %
CMT-Star CCM+ (hexahedral)	14.89	0.55	5.8 %
CMT-Star CCM+ (polyhedral)	15.04	0.56	7.7 %
UMass	14.37	0.53	1.9 %
Aachen	13.80	0.51	1.9 %
KAUST-Aramco	14.20	0.53	1.9 %
Siemens	13.55	0.50	3.8 %

Figure 12. Rate of Injection comparison for standard G (Non-flashing) condition

coming from the stationary simulations while the transient studies present similar differences claiming the same accuracy in the results.

Furthermore, the ROI prediction for the non flashing condition by hole 5 are compared between Argonne LES prediction, UMass, CMT, and Siemens, as shown in the Figure 13. It can be inferred the LES run predicts a higher ROI during the needle opening transient.

Figure 13. Rate of Injection comparison for standard G (Non-flashing) condition for hole 5

To further study the hole to hole variation in the ROI prediction, the relative standard deviation in the ROI predictions of each hole by CMT, UMass, Siemens are plotted in the Figure 14. The Relative Standard Deviation in the ROI across all 8 holes are calculated using the formulations represented in the Equation 40. In Equation 40 $\bar{m}$ and $\dot{m}_i$ refer to the instantaneous mass flow rate averaged over all 8 holes and the instantaneous mass flow rate through the hole numbered i , respectively.

$$RSD = \frac{100}{\bar{m}} * \sqrt{\frac{\sum_{i=1}^8 (\dot{m}_i - \bar{m})^2}{7}} \quad (40)$$

UMass observes higher hole to hole variations compared to CMT for dynamic mesh motion simulation. This difference in observation can be attributed to the difference in the turbulence closure model and different spatial discretization used by the two institutions. UMass has used a second order spatial discretization, whereas CMT has used a first order spatial discretization scheme. Hence, UMass observes more prominent transient vortices in their GDI simulations⁶. Siemens has also observed a transient hole to hole variation in their static mesh simulation, which is of similar magnitude compared to the CMT-CONVERGE simulation.

Figure 14. Relative Standard Deviation in Rate of Injection across all the holes for standard G (Non-flashing) condition

A similar analysis is also performed for the G2 flash boiling condition as seen in the Figure 15 and Figure 16. The ROI predicted for the flash boiling condition by the modelers are compared to the ROI measured under non flashing

condition due to the unavailability of experimental data for the flashing condition, and the ROI data are observed to be same for both the flash boiling and non-flashing conditions, which corroborates the hypothesis that these nozzles are being operated under chocking conditions.

Figure 15. Rate of Injection comparison for G2 (flash boiling) condition

Figure 16. Relative Standard Deviation in Rate of Injection across all the holes for G2 (flash boiling) condition

Higher hole to hole variation is observed for the flash boiling condition compared to the non-flashing condition. Towards the end of injection the variation in the ROI between the holes become prominent. This increase in variation can be attributed to the small value of mean ROI towards the EOI.

Momentum rate

For better understanding of the discharge through each hole, the momentum rate averaged over all the 8 holes are compared between the different institutions, for both the flashing and non-flashing conditions, and the Relative Standard Deviation across all the holes is also plotted. It is observed that modelers are predicting different momentum rate through different holes, though they match in their ROI prediction for both the flashing and non-flashing conditions,

as shown in the Figure 17 and Figure 18. Moreover, Table 6 provides the dimensionless flow coefficients for the transient cases as well as the data obtained from the CMT steady simulations compared to the experimental non-flashing results. It predicts once more, through the dimensionless coefficients, the existing differences between institutions due to significant deviation in the ROM results. Note that this difference between software in ROM prediction was observed by Payri et al.⁴¹ in a similar study. Figure 19 and Figure 20, which capture the hole to hole variation in momentum rate across all the 8 holes in non-flashing and flashing conditions respectively, are represented. Again, UMass observes a higher hole to hole variation in both the flashing and non-flashing conditions compared to CMT.

Figure 17. Momentum rate averaged over all 8 holes for the non-flashing condition

Figure 18. Momentum rate averaged over all 8 holes for the flashing condition

For a GDI moving needle simulation, the ROI remains constant when the needle is at the maximum needle lift resulting in a quasi steady state. Hence, the results for the transient moving needle simulations and the experimental observations are time averaged over the maximum needle lift duration. Argonne measures the projected mass density

Table 6. Comparison between experimental and computational nozzle flow coefficients in the steady state period (from 0.4 to 0.6 ms ASOI) for non-flashing condition.

	ROI (g/s)	ROM (N)	C_d	C_m	C_v	C_a
Experimental	13.82	2.63	0.52	0.40	0.73	0.75
CMT-CONVERGE	14.18	2.37	0.53	0.36	0.68	0.78
CMT-Star CCM+ (hexahedral)	14.89	2.86	0.55	0.43	0.78	0.71
CMT-Star CCM+ (polyhedral)	15.04	2.65	0.56	0.40	0.71	0.79
UMass	14.37	3.17	0.53	0.48	0.89	0.60

Figure 19. Relative Standard Deviation of Momentum rate across all 8 holes for the non-flashing condition

Figure 20. Relative Standard Deviation of Momentum rate across all 8 holes for the flashing condition

at a distance of 2 mm downstream away from the injector tip using an X-ray measurement technique. The measured quantity is time averaged over steady injection phase, which is similar to the time averaged fuel density simulated by the modelers.

The projected fuel mass density measured by Argonne across each hole is further compared with the UMass CFD prediction for fuel density under flash-boiling condition. Only UMass CFD predictions are chosen for the comparison, because the domain simulated by CMT is smaller and the effect of outlet is observed on the CFD predictions at 2 mm downstream.

Figure 21. Comparison of measured and predicted time-averaged fuel density across a line transecting holes 1 and 5 at 2 mm. Fuel density is defined as mass of fuel per volume of all species. Note the different axes.

Figure 21, clearly shows that the modelers are predicting very high fuel density compared to the experimental observations. The predicted plumes have a very high concentration of fuel as consequence of the difference in the momentum predictions, though the mass flow rate matches with the experiments.

The mass flow rate corresponds to the product of fuel density multiplied by a velocity, integrated over an area. The fact that the mass flow rates are well predicted by most of the CFD simulations indicates a good level of accuracy of these quantities within the nozzle, which is the choke point of the flow. If a quasi-stable flow is assumed, then the CFD codes should presumably be predicting the correct mass flow rate at the 2 mm observation line.

The mass flow rate constrains the product of density and velocity. Thus, the downstream over-prediction in density is indicative that the predicted velocity is far too low. This discrepancy in density predictions requires the modelers to revisit the mixing models of the non-condensable gas with the liquid fuel and the subsequent evaporation due to heat transfer.

Though no experimental data are available for Spray G and G2 droplet temperature, two computational predictions of temperature were compared to each other. These two were chosen because they reported temperature and because they used a sufficiently large domain. The domain for CMT is smaller compared to UMass, hence the comparison plane shown in Figures 22 and 23, was chosen at 1 mm away from the injector tip.

Figure 22. Predicted time averaged temperature field on a line across holes 1 and 5 located 1 mm downstream for the non-flashing condition.

Figure 23. Predicted time averaged temperature field on a line across holes 1 and 5 located 1 mm downstream for the flashing condition.

For the non-flashing Spray G conditions the temperature is largely dominated by the 573 K ambient, and the temperature predictions between the two codes are similar. Where the profile cuts through the spray, the temperature drops to near the 363 K fuel temperature. The predicted temperature profiles appear to be largely dominated by adiabatic mixing.

The predicted temperature profile for the flash boiling conditions for Spray G2 is more complex. The ambient temperature of 333 K is evident near the periphery of the spray. In the core, the CMT-CONVERGE predictions reach temperatures of 363 K, equal to the injected fuel temperature. The CONVERGE predictions of temperature seem to be largely driven by adiabatic mixing. The HRMFOam predictions show a relatively cool spray core, showing the effects of evaporative cooling. The veracity of the respective predictions is not yet determined, however experimental data for other flashing sprays have been reported. These data do suggest that evaporative cooling can depress droplet temperatures³³.

Figure 24. Predicted time averaged temperature field located at 1 mm downstream from the tip for the non-flashing condition by UMass.

Figure 25. Predicted time averaged temperature field located at 1 mm downstream from the tip for the flashing condition by UMass.

Figure 26. Predicted time averaged temperature field located at 1 mm downstream from the tip for the non-flashing condition by CMT using Converge.

Figure 27. Predicted time averaged temperature field located at 1 mm downstream from the tip for the flashing condition by CMT using Converge.

The aforementioned temperature predictions can further be corroborated by Figures 24, 25, 26, 27. Figures 24 and 25 represent the time averaged temperature contour at 1 mm downstream from the injector tip predicted by UMass for both the non-flashing and flashing conditions respectively, whereas Figures 26 and 27 represent the predictions by CMT using CONVERGE at the same location. Both the predictions look different visually. UMass observes a cooler spray core during the flashing conditions, which is not present in case of CMT. The size of the outlet plenum influences the temperature in the domain. Temperature in the periphery of the flow is not equal to the ambient temperature in the CMT-CONVERGE configuration whereas in the UMass calculation the value at the periphery is equal to the ambient value. Furthermore, a greater jet-to-jet interaction in terms of temperature is observed by CMT compared to UMass which could be due to the difference in the turbulence model.

Summary and Conclusions

The compilation of ROI predictions were, for the most part, very close to the experimental measurements. Surprisingly, the phase change model did not appear to be a major factor in predicting the mass flow rate. For example, RWTH Aachen's phase change model was based on equilibrium, yet gave predictions of mass flow rate nearly identical to the non-equilibrium HRM model used by all of the other codes. This result is consistent with the observations of Schmidt et al.³² that the mass flow rate through cavitating nozzles is consistent with assumptions of thermal equilibrium. This insensitivity to phase change model suggests that the success or failure of ROI prediction hinges more on numerical factors and simulation setup i.e. fuel properties than the phase change model. In further support of this assertion, we see that ROI predictions varied substantially among the users of the CCM+ code.

Even where predictions of mass flow rate were in close agreement, there were notable discrepancies in momentum flux at the hole exit. These differences persisted in both flashing and non-flashing conditions. This discrepancy may be due to numerical causes, since it was insensitive to the downstream condition. The comparisons indicate that different CFD codes, while using very similar models, can give very different velocity predictions for the external spray.

We also see an order of magnitude difference between the predicted and measured fuel concentration downstream of the injector. This large error likely indicates an issue with the fuel dispersion model. These Eulerian models of fuel/air mixing typically rely on the simplest possible closure, Fickian diffusion. This closure presumes that turbulent mixing is entirely responsible for the fuel-air mixing. Further, the fuel dispersion models are not cognizant of the large density ratio between phases.

Overall, the results show that modeling the internal flow is more advanced than the ability to simulate the spray plumes. While the ECN community was satisfied with a good success with ROI predictions, the external spray details were not consistently predicted. Hence, the external spray is a likely target for future research.

References

- Schmidt D, Gopalakrishnan S, and Jasak H,(2010) Multi-dimensional simulation of thermal non-equilibrium channel flow. *International Journal of Multiphase Flow*,36-4:284-292.
- Gopalakrishnan S and Schmidt DP,(2008),A computational study of flashing flow in fuel injector nozzles *SAE International Journal of Engines*.
- Neroorkar K, Schmidt D.,(2011), A Computational Investigation of Flash-Boiling Multi-hole Injectors with Gasoline-Ethanol Blends,*SAE Technical Paper*,2011-01-0384.
- Neroorkar K,Schmidt D. ,(2011), Modeling of vapor-liquid equilibrium of gasoline-ethanol blended fuels for flash boiling simulations,*Fuel*
- Neroorkar K,Shields B,Grover Jr R O,Torres, A P, Schmidt D.,(2012), Application of the homogeneous relaxation model to simulate cavitating flow of a diesel fuel,*SAE Technical Paper* , 2012-01-1269.
- Baldwin E.T. , Grover R.O. , Parrish S.E. , Duke D.J. , Matusik K.E. , Powell C.F. , Kastengren A.L. , Schmidt D.P. ,(2016),String flash-boiling in gasoline direct injection simulations with transient needle motion,*International Journal of Multiphase Flow*
- Moulai M and Grover R and 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*,2015-01-0944.
- Colarossi M,Trask N,Schmidt D,bergander M ,(2012), Multidimensional modeling of condensing two-phase ejector flow,*International Journal of Refrigeration*.
- Strek P. , Duke D. , Swantek A. , Kastengren A. , Powell C.F. , Schmidt D.P. (2016), X-Ray radiography and CFD studies of the spray G injector, *SAE Technical Paper* , 2016-01-0858.
- Duke D, Swantek A, Tilocco Z, Kastengren A,Fezzaa K,Neroorkar K, Moulai M, Powell C and Schmidt D, (2014),X-ray imaging of cavitation in diesel injectors,*SAE International Journal of Engines*.
- Mohapatra C, Jacobsohn G, Baldwin E, Schmidt D, (2017) Modeling sealing in transient injector simulations. *ASME 2017 Fluids Engineering Division Summer Meeting 2017*.
- Desantes J M and Payri R and Gimeno J and Marti-Aldaravi P,(2016),Coupled/decoupled spray simulation comparison of the ECN spray A condition with the Σ -Y Eulerian atomization model.*International Journal of Multiphase Flow*.
- Bilicki Z and Kestin J, Physical aspects of the relaxation model in two-phase flow,(1990)*Proceedings of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*.
- Lemmon EW , Huber ML and McLinden MO, (2013), NIST standard reference database23: reference fluid thermodynamic and transport properties-REFPROP, Version 9.1, NIST,*Gaithersburgh. MD*.
- Downar-Zapolski P, Bilicki Z , Bolle L and Franco J, (1996) The non-equilibrium relaxation model for one-dimensional flashing liquid flow.*International Journal of Multiphase Flow*.
- Battistoni M. , Duke D.J.,Swantek A.B. , Tilocco F.Z. , Powell C.F.,Som S., (2015),Effects of noncondensable gas on cavitating nozzles,*Atomization and Sprays*.
- Saha K. , Som S. and Battistoni M.,(2016),Parametric Study of HRM for Gasoline Sprays,*ILASS Americas 28th Annual Conference on Liquid Atomization and Spray Systems, Dearborn, MI, May 2016*

18. Yakhot V and Orszag S, (1986), Renormalization group analysis of turbulence. I. Basic theory, *Journal of scientific computing*
19. Menter F.R. (1993), Zonal Two Equation k- ω , Turbulence Models for Aerodynamic Flows, *AIAA paper*.
20. Gueyffier D., Li J., Nadim A., Scardovelli R., Zaleski S., (1999), Volume-Of-Fluid interface tracking with smoothed surface stress methods for three dimensional flows, *Journal of Computational Physics*.
21. Richards K.J., Senecal P.K., and Pomraning E.(2016), CONVERGE v2.3 Documentation, *Convergent Sciences Inc*.
22. Rider W.J., Kothe D.B., (1998), Reconstructing Volume Tracking, *Journal of Computational Physics*
23. Pomraning E., and Rutland C.J., (2002), Dynamic one-equation non-viscosity large-eddy simulation model, *AIAA Journal*
24. Bode M., Deshmukh A.Y., Falkenstein T., Colonius T., and Pitsch H., (2019), Hybrid scheme for complex flows on staggered grids and application to multiphase flows, *Journal of Computational Physics*
25. Bode M., Satcunanathan K., Colonius T., and Pitsch H., (2018), An Equation of State Tabulation Approach for Injectors with Non-Condensable Gases: Development and Analysis, *Proceedings of the 10th International Symposium on Cavitation (CAV2018)*
26. Bode M., Falkenstein T., Le Chenadec V., Kang S., Pitsch H., Arima T., and Taniguchi H., A new Euler/Lagrange approach for multiphase simulations of a multi-hole GDI injector, *SAE technical Paper*, 2015-01-0949.
27. Bode M., Falkenstein T., Davidovic M., Pitsch H., Taniguchi H., Murayama K., Arima T., Moon S., Wang J., Arioka A., (2017), Effects of Cavitation and Hydraulic Flip in 3-Hole GDI Injectors, *SAE International Journal of Fuels and Lubricants*.
28. Bode M., Davidovic M., Pitsch H., (2019), Towards Clean Propulsion with Synthetic Fuels: Computational Aspects and Analysis, *High Performance Computing in Science and Engineering '18*.
29. Di Napoli E., Hermans M.A., Iliev H., Lintermann A., and Peyser A., (2017), Multi-scale Coupling for Predictive Injector Simulations, *High-Performance Scientific Computing*.
30. Matusik K., Duke D., Sovis N., Swantek A., Powell C., Payri R., Vaquerizo D., Giraldo Valderrama J.S., Kastengren A., (2017), A study on the relationship between internal nozzle geometry and injected mass distribution of eight ECN Spray G nozzles, *InIlass Europe. 28th european conference on Liquid Atomization and Spray Systems 2017*
31. Yue Z., Battistoni M., Som S., (2018), A numerical study on spray characteristics at start of injection for gasoline direct injection, *InICLASS Chicago, 14th triennial international conference on liquid atomization and spray systems 2018*
32. Schmidt, D and Rakshit, Sukanta and Neroorkar, Kshitij, (2009), Thermal and inertial equilibrium in small, high-speed, cavitating nozzle simulations, *11th International Conference on Liquid Atomization and Spray Systems (ICLASS-2009)*, Vail, CO, July
33. Zhang G., Hung D. and Xu M., (2014), Experimental study of flash boiling spray vaporization through quantitative vapor concentration and liquid temperature measurements, *Experiments in fluids*
34. Kastengren A. and Powell C. F., (2014), Synchrotron X-ray techniques for fluid dynamics, *Experiments in fluids*
35. Kastengren A.L., Tilocco F.Z., Duke D. and Powell C.F., (2014), Time-resolved X-Ray radiography of sprays from Engine Combustion Network spray A diesel injectors, *Atomization and sprays*
36. Kirkpatrick P. and Baez A.V., (1948), Formation of Optical Images by X-Rays, *J. Opt. Soc. Am.*
37. Payri R., Gimeno J., Marti-Aldaravi P. and Vaquerizo D. (2016). Internal flow characterization on an ECN GDI injector. *Atomization and Sprays*, 26(9), 889919.
38. Routson D., Ferguson J., Crepeau J., McEligot D., Budwig R. (2013). Reynolds averaged navier stokes models compared to direct numerical simulations in an adverse pressure gradient boundary layer over a flat plate. *American Society of Mechanical Engineers, Fluids Engineering Division (Publication) FEDSM*, 1 A(May 2016).
39. Payri R., Gimeno J., Marti-Aldarav P., Vaquerizo D. (2016). Internal flow characterization on an ECN GDI injector. *Atomization and Sprays*, 26(9), 889919. <https://doi.org/10.1615/AtomizSpr.2015013930>
40. Engine Combustion Network (ECN). (n.d.). Retrieved from <https://ecn.sandia.gov/gasoline-spray-combustion/>
41. Payri R., Gimeno J., Marti-aladaravi P., Martinez M. (2019). Nozzle Flow Simulation of GDI for Measuring Near-Field Spray Angle and Plume Direction. *SAE Technical Paper 2019-01-0280*, 111.