

A Computational Study of Flashing Flow in Fuel Injector Nozzles

S.Gopalakrishnan and David P. Schmidt
University of Massachusetts – Amherst, USA

Copyright © 2008 SAE International

ABSTRACT

Flash boiling conditions, where the fuel is superheated with respect to cylinder pressure, are often found in gasoline direct injection engines. This phenomenon affects the flowrate of the fuel and can cause choking of the nozzle. In this work we present multi-dimensional simulations of flashing internal injector flow. The modeled fluid quality (mass fraction of vapor) tends towards the equilibrium quality based on the Homogenous Relaxation Model. The relaxation time is dependent on the local pressure, the vapor pressure, and the void fraction. Simulations of the internal flow are presented and, where possible, validated with experimental data. Both two- and three- dimensional computational results show geometrically-induced phase change, similar to cavitation, near the nozzle entrance. Near the nozzle exit plane the phase change occurs at all radial locations and can be quite sensitive to temperature. Three-dimensional simulations run in an asymmetric injector tip at higher injection pressure show reduced sensitivity to temperature.

INTRODUCTION

The rise in popularity of direct injection systems for gasoline engines necessitates the understanding of the complex phenomena of the fuel spray and its vaporization. The spray structure in the combustion chamber is affected by the external breakup of the jet which itself depends upon the internal flow characteristics in the fuel injector nozzle. The fuel in the injection system can acquire heat from the relatively hot surroundings and from compression during pumping, raising its temperature and the vapor pressure. If the pressure downstream of the injector is less than the vapor pressure, the fuel will likely flash boil. Though primarily associated with gasoline direct injection [1], recent work has suggested that flash-boiling could play a role in premixed-charge compression ignition

engines with special fuels [2].

Flash boiling is a phenomenon similar to cavitation, which is also known to occur in gasoline injectors [3]. Both are a transition from liquid to vapor due to a drop in pressure. In contrast to nucleate boiling, the enthalpy for vaporization is not provided at walls during the phase change process, but is instead provided by inter-phase heat transfer. A gross distinction between cavitating and flash-boiling nozzle flow is simply that the enthalpy of the cavitating flow is below the saturated liquid enthalpy at the downstream pressure, while the enthalpy of the flash-boiling flow exceeds the saturated liquid enthalpy at the downstream pressure. A detailed discussion of the differences must be deferred until after some review of phase change physics.

An experimental study by Oza and Sinnamon [4] found that flashing occurred in two modes, namely the “external flashing mode” and the “internal flashing mode.” Park and Lee [5] performed investigations using transparent nozzles and reported the flashing modes in the internal and the external flow. Apart from the degree of superheat it was found that the factors affecting flash boiling included the nozzle geometry, surface roughness, turbulence and physical properties of the fuel. The external spray characteristics as result of flash boiling of superheated fuel have been studied experimentally by several researchers [6, 7, 8].

Even though only a very small fraction of the liquid mass changes phase while still in the nozzle, this small amount of mass can occupy a large volume and greatly affect the nozzle flow. However, there are many investigations of external sprays under flash-boiling conditions and relatively few studies of the internal flashing flow. As noted by Park and Lee [5], the phase change process definitely begins within the nozzle. Experimental studies of flashing nozzle flow have primarily been conducted with wa-

ter, such as Reitz [9] and Fauske [10]. Fauske noted that the flow through nozzles will choke under saturated conditions (with respect to the upstream conditions) and low downstream pressure.

There have been a few attempts to model the internal flashing flow, almost all limited to one-dimensional flow. For a recent example of one-dimensional modeling, the reader may consult Barret et al. [11]. Such models are often not satisfactory for the low length-to-diameter ratios used in most fuel injector nozzles, where the nozzle flow is separated and displays two and three-dimensional features. Schmidt et al. [12] reported the development of a two-dimensional flashing simulation for simple nozzle shapes, which offers some insights into short flashing-nozzle flow. However, Schmidt et al. were limited to simple, two-dimensional block-structured meshes. This initial effort produced a code that was not terribly robust or efficient. Fortunately, the numerical techniques available and the understanding of the special requirements of modeling flashing flow have progressed.

Any numerical investigation in the physics of the atomization consists of two distinct sections, the external jet break up and the internal nozzle flow. The calculations of the nozzle flow calculations serve as inputs to external jet break up models. The external modeling of flash boiling sprays has been presented in the past by Zeng et al. [13], Zuo et al [14], and Kawano et al. [15].

These external spray models have been developed with the best information available, but the open literature has very little to offer about the details of the internal flashing flow. Important questions remain about the velocity of the fluid leaving the nozzle and the fraction of vapor present at the nozzle exit. Further, we wish to know more about how nozzle geometry and injection pressure affect the internal flow. The goal of this paper is to present a multi-dimensional CFD model capable of simulating flash-boiling flow.

The foremost question in a CFD simulation of flash boiling is how to model the heat transfer between the two phases. The rate of heat transfer between the two phases can limit the phase change and is invariably dependent on the temperature of the liquid. This role of heat transfer may be contrasted to cavitating flow. For cavitating flows [16] the temperature of the entering fluid is generally quite low and the energy transfer from the liquid phase to the vapor is consequently lower. The timescale of the heat transfer is several orders of magnitude lower than that of the flow-through time. Thus, assumptions of thermal equilibrium in cavitation models were quite successful in repeated modeling efforts [17] [18] [19].

If the enthalpy of the fluid is high, then the phase change is akin to a boiling process. The majority of the enthalpy needed for vaporization in the flash boiling mechanism is given by way of inter-phase heat transfer. As temperature increases, the saturated vapor density increases as

the saturated liquid density decreases. As a result of this trend, the amount energy to be provided by the liquid per unit volume of vapor generated is much higher than in typical cavitating flow despite the decrease in the enthalpy of vaporization. The dimensional analysis of Kato et al.[20] can help identify the relative magnitude of inertial versus thermal rates in phase change.

Kato et al. [20] analytically studied similar thermal effects which limited the growth rate of bubbles in liquids. They calculated rate of formation of vapor as a function of the energy transfer at the interface. Kato et al. found that two non-dimensional parameters, the Jakob number, Ja , and non-dimensional time t^* played an important role in the vaporization. The Jakob number is defined as,

$$Ja = \frac{\rho_l c_p \Delta T}{\rho_v h_{fg}} \quad (1)$$

where ΔT is the degree of superheat and h_{fg} is latent heat of vaporization. The Jakob number is ratio of the amount sensible heat available to the amount of energy required for vaporization. The non-dimensional time is given by the following expression, where k is the thermal diffusivity of the liquid.

$$t^* = \frac{t \Delta P}{k \rho_l} \quad (2)$$

The study revealed that for large Jakob number and non-dimensional time, the inertial effects dominate and vice versa when they are small, the physics are mainly guided by thermal effects. In processes where cavitation occurs the Ja and t^* are quite large, whereas in problems with flash boiling involved they are significantly smaller such that thermal effects need to be considered. This necessitates the use of a finite rate heat transfer process where the system is not in thermodynamic equilibrium but rather relaxes to it over a fixed time.

From a computational perspective the problem can be approached with two methods. The first is where the two phases are tracked explicitly as separate fluids. The second approach entails the use of a pseudo-fluid where the density varies continuously over the computational domain and the phases are quantitatively known using the mass fraction. The advantage of the pseudo-fluid approach is that it is relatively easier to implement, and solutions can be obtained rather quickly as compared to the more general explicit approach.

The explicit approach would require extremely tight coupling between the phase velocities and interphase drag models, which will lead to problems such as higher computational cost and numerical instabilities. The latter approach conveniently avoids these issues and if sufficiently resolved will permit relative velocities between the phases on large scales, i.e. a liquid core with annular flow of va-

por. The pseudo–fluid approach is preferred here for simplicity, computational efficiency, and stability.

The primary casualty of this assumption is the ability to capture the slip velocity between the vapor and liquid. Here, the fact that this is a multi-dimensional study is very important. Whereas a one-dimensional velocity without interphase slip is often inadequate, in two and three-dimensions the velocity difference between gas and liquid can still be captured in a macroscopic sense. For example, in two-dimensions an annular flow with vapor around the outside can be modeled. The CFD simulation will likely predict a distinct velocity difference between the outside vapor region and the liquid core. Thus, inertial interphase equilibrium in higher-dimensions is a much less restrictive assumption than in one-dimension.

In this study we use the pseudo–fluid approach combined with the Homogeneous Relaxation Model (HRM) to model flashing flow in fuel injectors. Previous studies using the Homogeneous Relaxation Model have been done in one dimension [21]. A non–equilibrium flow model for two dimensions has been presented by Schmidt et al [12]. Here we present a complete two and three dimensional implementation of the model with support for polyhedral meshes. Parallelization using message-passing (MPI) and more advanced numerical methods are presented as well.

GOVERNING EQUATIONS

The governing equations reflect the general philosophy of relying on the basic conservation equations as much as possible, and then relegating the nearly intractable physics of non-equilibrium phase change to empiricism. The challenge is finding a suitable empiricism and determining how this relationship can be connected with the conservation equations. One possible approach is the Homogeneous Relaxation Model, represented by Eqn. 3, which determines the substantial derivative of the mass fraction of vapor. In the remainder of the text, the mass fraction of vapor will be referred to, in short, as "quality."

$$\frac{Dx}{Dt} = \frac{\bar{x} - x}{\Theta} \quad (3)$$

In Eqn. 3, x is the instantaneous quality which exponentially relaxes to the equilibrium quality $\bar{x}$ over a timescale of Θ . Eqn. 3 is based on the linearized expansion proposed by Bilicki and Kestin [22]. The idea behind this model is that the quality of the mixture will return to the equilibrium value according to some complicated function that can be linearized. The generalized model form itself originated from the work done by Einstein [23] in refrigeration modeling and has been employed by several others to model one dimensional two phase flow. The timescale Θ can be obtained from the correlations for relaxation time as suggested by Downar–Zapolski *et al* [21]. In the present work, a combination of two suggested cor-

relations is used. In order to include the effects of interfacial area and a tendency to move towards the saturation pressure, the time scale depends on a the void fraction, α , and a non-dimensional form of pressure, ϕ . For all the results shown here, $\phi = \frac{P_{sat} - P}{P_{sat}}$ and the value of Θ_0 is $6.51 \cdot 10^{-4} [s]$. The timescale employed in Eqn. 3 is calculated from Eqn. 4.

$$\Theta = \Theta_0 \alpha^{-0.54} \phi^{-1.76} \quad (4)$$

The continuity and momentum equations, Eqn. 5 and Eqn. 6 respectively, are combined together with Eqn. 3 to achieve close coupling and to obtain an equation for pressure. The mass flux is given by ϕ in Eqn. 5 and Eqn. 6 and the stress tensor is denoted by $\bar{\tau}$. Currently no turbulence models have been implemented and laminar flow is assumed. A simple constant value is used for viscosity; on-going work is investigating different models for viscosity.

$$\frac{\partial \rho}{\partial t} + \nabla \phi = 0 \quad (5)$$

$$\frac{\partial \rho U}{\partial t} + \nabla(\phi U) = -\nabla p + \nabla \bar{\tau} \quad (6)$$

The discretized form of the momentum equation is represented in Eqn. 7 by employing the notation used by Peric and Ferziger [24]. The subscript p denotes the computational cell of interest and the H operator represents the off diagonal coefficients of the linear system of equations and the variable a_p is the diagonal term.

$$a_p U_p = H(U) - \nabla p \quad (7)$$

When the system is not in thermodynamic equilibrium the density becomes a function of three independent variables, i.e. pressure, enthalpy and quality. The substantial derivative of density is given by the following expression.

$$\frac{D\rho}{Dt} = \frac{\partial \rho}{\partial p}|_{x,h} \frac{Dp}{Dt} + \frac{\partial \rho}{\partial x}|_{p,h} \frac{Dx}{Dt} + \frac{\partial \rho}{\partial h}|_{x,p} \frac{Dh}{Dt} \quad (8)$$

Eqn. 9, which gives us the velocity divergence at the new time step, is obtained by subtracting the conservation of mass Eqn. 5 from Eqn. 8. The enthalpy currently is assumed to be constant and hence the last term in Eqn. 8 is neglected.

$$-\rho \nabla \cdot U = \frac{\partial \rho}{\partial p}|_{x,h} \frac{Dp}{Dt} + \frac{\partial \rho}{\partial x}|_{p,h} \frac{Dx}{Dt} \quad (9)$$

An expression for the velocity divergence can be derived by dividing Eqn. 7 by the diagonal coefficient a_p and then

taking its divergence. By substituting this in the above equation and rearranging the terms.

$$\frac{\partial \rho}{\partial p}|_{x,h} \frac{\partial p}{\partial t} + \frac{\partial \rho}{\partial p}|_{x,h} (U \cdot \nabla p) + \rho \nabla \cdot \left(\frac{H}{a_p} \right) - \rho \nabla \frac{1}{a_p} \nabla p + \frac{\partial \rho}{\partial x}|_{p,h} \frac{Dx}{Dt} = 0 \quad (10)$$

The addition of the product of p and Eqn. 5 with Eqn. 10 results in Eqn. 11 in which the velocity is now formulated as the volumetric flux, ϕ_v .

$$\frac{1}{\rho} \frac{\partial \rho}{\partial p}|_{x,h} \left(\frac{\partial \rho p}{\partial t} + \nabla \rho p \phi_v \right) + \rho \nabla \cdot \left(\frac{H}{a_p} \right) - \rho \nabla \frac{1}{a_p} \nabla p + \frac{\partial \rho}{\partial x}|_{p,h} \frac{Dx}{Dt} = 0 \quad (11)$$

The main features of this equation are that the terms containing pressure are linear and the HRM model can be conveniently implemented in the last term. The transmission of pressure waves which is essential for any compressible flow calculation is allowed by the transient and convective terms in the equation while the pressure is kept in range and is damped by the Laplacian term. For low Mach number flows the terms containing $\frac{\partial \rho}{\partial p}|_{x,h}$ can be dropped. The terms were retained for the two-dimensional calculations but dropped for the three-dimensional calculations since they change mass flow rate very little but slow the rate of solver convergence.

A predictor–corrector approach similar to the PISO method [25], is used to solve the set of equations. At a given time step the velocity field is first computed using the lagged pressure. Multiple PISO iterations are then used to obtain the correct pressure and subsequently the velocity field is updated with the corrected flux.

NUMERICAL METHOD

The first step is to solve the continuity equation implicitly.

$$\frac{\partial \rho}{\partial t} + \nabla(\phi_v \rho) = 0 \quad (12)$$

The volumetric flux ϕ_v is computed from the previous time step to obtain the new density. The current value of the flux ϕ is calculated based on this value. All the thermodynamic variables such as compressibility, quality void fraction etc are now obtained.

The velocity is then computed from the momentum equation. This velocity U^0 is based on the lagged pressure p^n and is corrected later using the updated pressure.

$$\frac{\partial \rho U^0}{\partial t} + \nabla(\phi U^0) = -\nabla p^n + \nabla \mu \nabla U^0 \quad (13)$$

The velocity field prior to the pressure projection can be represented as the ratio of off–diagonal terms to diagonal terms in Eqn. 7. This is used to compute ϕ^* which is used in Eqn. 12.

$$U^* = \frac{H}{a_p} \quad (14)$$

The phase change term in Eqn.11 can be expressed using the phase change model introduced in Eqn. 3 and the partial derivative from the chain rule. The phase change term is denoted as M , for short.

$$M \equiv \frac{\partial \rho}{\partial x}|_{p,h} \left(\frac{\bar{x} - x}{\Theta} \right) \quad (15)$$

The term M is highly dependent on pressure and is non–linear. The dependence occurs because of the strong effect of pressure in the timescale in Eqn. 3 and because of the temperature dependence of properties. In order to linearize this term the PISO iterations are used in conjunction with secant iterations. The current PISO iteration is denoted by the superscript $k+1$ and the previous iteration is k .

$$\rho \nabla \cdot \left(\frac{H}{a_p} \right) - \rho \nabla \frac{1}{a_p} \nabla p^{k+1} + M(p^k) + \frac{\partial M}{\partial p}(p^{k+1} - p^k) = 0 \quad (16)$$

Eqn. 16 is solved iteratively around 5 times. The pressure field at the end of the iterations is used to correct fluxes at the current timestep.

SOFTWARE FRAMEWORK

This numerical model is implemented using the open-source OpenFOAM [26] libraries for continuum mechanics. OpenFOAM provides a convenient framework in the object oriented paradigm which is suited for rapid construction of CFD codes. It provides support for polyhedral meshes and gives us choice from a wide variety of discretization schemes. Parallelization can be easily done using this set of libraries.

RESULTS

AXISYMMETRIC NOZZLE An experimental investigation using an axisymmetric nozzle and water as the working fluid was done by Reitz [9]. The downstream pressure was kept at 101 kPa and the inlet pressure was fixed at 787 kPa. He varied the inlet temperature and noted the effect on mass flow rate. The mass flow rate gradually decreased until an upstream temperature of about 430K was reached, whereupon the mass flow discontinuously dropped. Reitz reported that the mass flow rate dropped below the measurement range of his flow meter.