ICLASS 2009, 11th Triennial International Conference on Liquid Atomization and Spray Systems, Vail, Colorado USA, July 2009

Thermal and Inertial Equilibrium in Small, High-Speed, Cavitating Nozzle Simulations

David P. Schmidt* Sukanta Rakshit, and Kshitij Neroorkar
Department of Mechanical and Industrial Engineering
University of Massachusetts
Amherst, MA 01003 USA

Abstract

The construction of any simulation of cavitating injector nozzles begins with the fundamental assumptions of which phenomena will be included and which will be neglected. To date, there has been no consensus about whether it is acceptable to assume that small, high speed cavitating nozzles are in thermal or inertial equilibrium. This diversity of opinions leads to a variety of modeling approaches. If one assumes that the nozzle is in thermal equilibrium, then there is presumably no significant delay in bubble growth or collapse due to heat transfer. Heat transfer is infinitely fast and phase change is limited by inertial effects. The assumption of inertial equilibrium means that the two phases have negligible slip velocity. Alternatively, on the sub-grid scale level, one may also consider the possibility of small bubbles whose size responds to changes in pressure. The present work uses experimental results collected from the open literature to evaluate assumptions that have been used in previous modeling results. Multidimensional CFD results that assume thermal equilibrium and non-thermal equilibrium are used to further evaluate these particular assumptions with plain orifice injector nozzles. The results indicate that equilibrium assumptions are sufficient to predict mass flow rate and cavitation incidence in small, high-speed nozzle flows. Effects of thermal non-equilibrium are not evident in experimental measurements of mass flow rate.

Introduction

Simulations of cavitating atomizer nozzles invariably require simplifying assumptions. These assumptions should be sufficient to render the problem tractable without producing unacceptable errors. The following paper examines common assumptions that may be made in cavitating nozzle simulations and estimates what their impact is on the simulation fidelity.

In the current context of atomization, the goal of the cavitation simulations would be to provide the critical information for understanding and predicting the downstream atomization. The foremost characteristic of a nozzle's performance is whether the nozzle is experiencing hydraulic flip, where downstream, non-condensable gas is entrained into the nozzle [1]. For non-axisymmetric nozzles, the possibility of partial hydraulic flip must also be considered. Hydraulic flip dramatically suppresses atomization, but is not actually a form of cavitation, and so is beyond the purview of the current discussion.

The most important single metric of cavitating nozzle flow is the mass flow rate, or alternatively, coefficient of discharge. Nearly every atomization application requires accurate knowledge of how much mass is atomized. After mass flux, the momentum flux is the second most important feature of a cavitating nozzle flow. The momentum flux correlates closely with gas entrainment and spray penetration [2]. The next major descriptor of a nozzle's performance is the disturbance level which it imposes on the exiting flow. The cavitating nozzle may eject bubbles or demonstrate transient film oscillation [3][4] that greatly perturb the spray. Other characteristics of nozzle performance that can affect the spray are the exit velocity profile and transverse velocity magnitude. Predicting all of these characteristics, which are listed in order of importance, represent the goals of cavitation simulations.

Past Work

Past efforts to model cavitating nozzles have taken different approaches with differing assumptions. One of the earliest approaches was to use bubble dynamics equation. This was done by solving the Rayleigh-Plesset equation to get the necessary information for the growth and destruction of the bubble. Kubota [5] in their cavitation Bubble Two-phase (BTF) Model assumed that the inside and outside of the cavity were a continuum. He also considered the cavity as a compressible viscous fluid whose density varied extremely. A similar kind of approach was used more recently by Chen and Heister [6] in which hydrodynamic non-equilibrium and bubble dynamics are of primary concern. Using their approach they showed that earlier approach was valid only when bubble radius is very small. When the bubble radius is comparable to the distance between the two bubbles, one cannot assume the local pressure as far

*Corresponding author, schmidt@ecs.umass.edu

field pressure. Chen and Heister averaged the local pressure and solved the Rayleigh-Plesset equation for a finite domain. From this approach they related the pseudo-density to the average pressure and void fraction from mass balance considerations.

Later Gavaises and Arcoumanis [7] in their work employed a Lagrangian model to represent the bubble growth. They extended the transport model to include the bubble breakup and related events. Alebjecvic [8] introduced a cavitating flow using a two fluid model with a continuous liquid and a dispersed vapor phase. The model is based on linearized Rayleigh-Plesset equation which is driven by difference between hydrodynamic and vapor pressures. In his model, he also used an additional corrector term to account turbulence pressure fluctuations. With this model he evaluated important spray characteristics and cavitation flow in nozzles.

Chen and Heister [9] also worked out a model where they treated the zone of cavitation as two phase mixture which they used for two phase modeling. They solved the continuity equation postulating that the change in density must be related to the pressure difference $P-P_v$.

Avva et al. [10] introduced an enthalpy based model for two phase cavitation modeling which minimizes the need for empiricism. In this model they assumed local thermodynamic equilibrium between the liquid and vapor phases, but allowed mechanical non-equilibrium such as unequal velocities or velocity. They formulated a single energy equation for a homogeneous mixture of gases which computes the enthalpy as dependent variable. The volume fraction and other thermodynamics properties were computed using the enthalpy.

In a third kind approach for two phase modeling was employed by Delannoy and Kueny [11] where an equation of state is derived based on a barotropic model between pressure and density to predict the flow in a venturi. This model assumed thermodynamic as well as mechanical equilibrium between the two phases. They successfully demonstrated the qualitative behavior of the venture but fail to employ the frequency of bubbles or the length. Schmidt et al. [12] later developed a two dimensional transient model which was intended for small, high speed nozzle flows, such as in diesel fuel injectors. This model also used a barotropic equation of state but included the compressibility of both the liquid and the vapor phase. The pressure-density relationship was derived by integrating the result of the Homogenous Equilibrium Model for the speed of sound. Habchi and Dumont [13] used HEM to simulate the cavitating flow inside a diesel multi-hole injector. In their work they successfully simulate a viscous, unsteady cavitating flow inside a diesel fuel injector.

The goal of the present work is to explore the impact of the assumptions of hydrodynamic and thermal non-equilibrium on the prediction of the specific nozzle characteristics listed above.

Inertial Equilibrium

In the most general sense, liquid and vapor velocity are two completely separate fields, coupled by inter-phase drag. Alejobevic et al. [8] used the two-velocity formulation to represent the separate vapor and liquid velocities. A more common approach is to assume that the vapor moves as bubbles transported by the liquid [7]. The simplest assumption is have both phases move at the same velocity. This assumption of negligible slip is equivalent to an inter-phase drag that is so large, that inter-phase slip is negligible.

The computation of separate velocity fields is, in some sense, the most general modeling assumption. However, tracking the vapor's velocity, separate from the liquid, may not actually be useful. As mentioned in the introduction of this paper, the purpose of modeling cavitating flow in nozzles is to understand and predict the influence on the downstream spray. It is questionable whether calculating the separate velocity field is helpful in predicting this influence.

The reason why the vapor phase's velocity is not important is because of the extreme density ratio between vapor and liquid. At room-temperature, the density of saturated liquid is several times that of the vapor density of most commonly used fluids. This means that the vapor flow does not represent a significant amount of mass. Since momentum is the product of mass times velocity, the momentum error is not significant. Quite simply, the vapor makes no contribution to the spray, which is entirely liquid. The presence of the vapor represents a cavity, devoid of the liquid that produces the spray.

It is also difficult to justify the application of Volume-of-Fluids (VOF) to simulation cavitating orifices. These nozzles are extremely high Weber number, where the role of surface tension on the resolvable scales is minimal. There is also no expectation that the interface must be smooth. Experimental evidence, such as from Schmidt et al. [4] shown below in Figure 1 indicate that the interface is indeed smooth in the favorable pressure gradient just after separation off the inlet corner, but in the remainder of the nozzle, the surface is quite convoluted. The application of VOF would artificially smooth the entire interface, at considerable cost with little benefit.

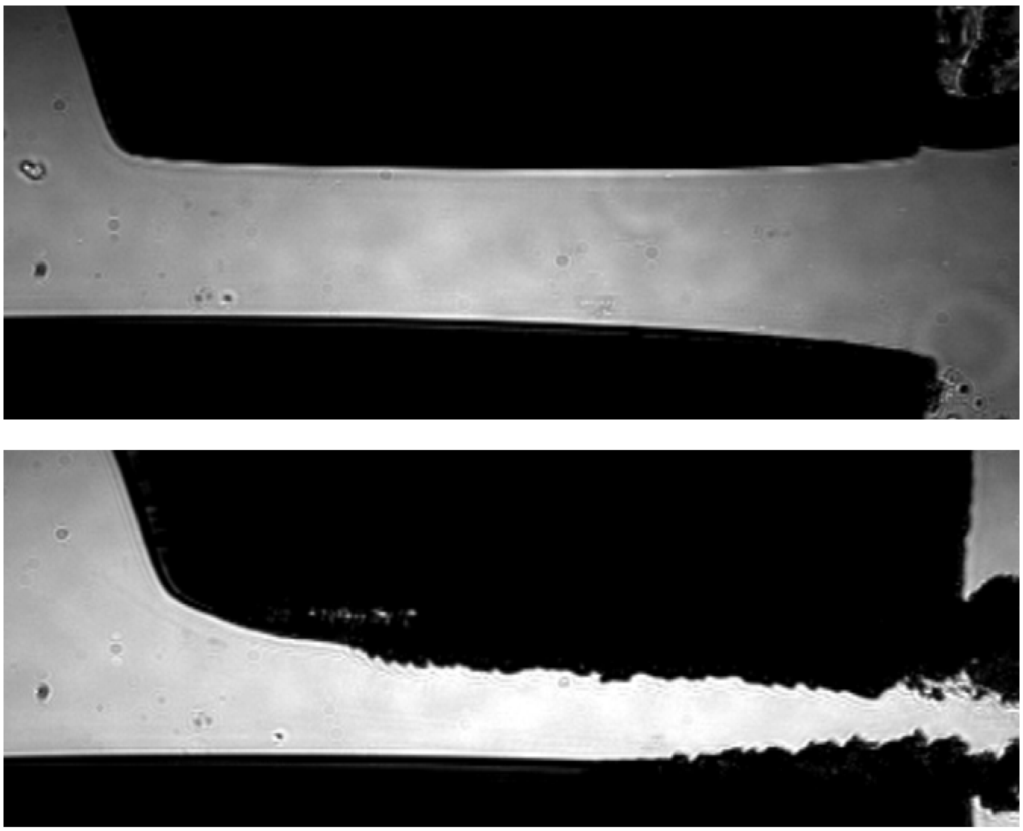

Figure 1. The nozzle shape, shown above in a non-cavitating condition and below, a back-lit view of cavitation. Flow is from left to right through a two-dimensional planar nozzle [4].

The assumption that there is a field of small-scale bubbles also does not agree perfectly with images by Arcoumanis et al. and others. However, there are practical advantages to the bubble-transport approach. The foremost is that cavitation damage is likely governed by small-scale features of bubble collapse. The challenge is that collapsing bubbles at a wall are not spherical and may be affected by neighboring bubbles, and thus the standard Rayleigh-Plesset model is not ideal.

Thermal Equilibrium

Phase change, including cavitation, may be in thermal non-equilibrium. Kato et al. [14] analyzed the relative importance between the inertial and thermal effects for bubble growth. They concluded that the radius of the inertial bubble grew linearly with time, whereas the radius of the thermal bubble grew with the square root of time and so the inertial effects will initially dominate the bubble growth. It is seen from their conclusions that the length scales required for the thermal effects to be important are much larger than the nozzles usually used in injectors. The reason for this is that the majority of the energy required for phase change is provided by the inter-phase heat transfer. In cavitating flows, the heat transfer is extremely fast [15] and hence the time scale for heat transfer is much smaller than the flow through times in the nozzle. However, this is not the case in hot fluids, in which the rate of heat transfer is limited by the inter-phase heat transfer process.

The Jakob number is a non-dimensional number and can be used to understand the importance of considering the thermal non-equilibrium effects. This number is calculated from Eqn. 1 below.

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

Here ρ is the density and the subscripts l and v represent liquid and vapor, respectively. The variable C_p stands for the specific heat at constant pressure, ΔT represents the degree of superheat, and h_{fg} is the latent heat of vaporization. The Jakob number is the ratio of the amount of sensible heat available to the amount of energy required for vaporization. As the temperature of the fluid increases, the denominator of the above equation increases. For example, consider water at two temperatures: 25degC and 50degC. The value of $\rho_v h_{fg}$ at these two temperatures is 56.32 KJ/m and 198.06 KJ/m, respectively. This shows that, as the temperature increases, the amount of energy required for vaporization increases and hence the time required for the inter-phase heat transfer will be higher and could be comparable to the flow through times in the nozzle. In such cases ignoring the thermal effects can lead to inaccuracies. Schmidt [12] simulated the flash-boiling experiment of Reitz [16] using a cavitation model that was based on the assumption of thermal equilibrium and another model that was based on non-equilibrium. It was observed that as the temperature of the fluid was increased, the mass flow rates predicted by the equilibrium model diverged further from the experimental values; however, the non-equilibrium model successfully predicted the experimental mass flow rates.

Nurick [17] developed a simple and elegant model to explain the behavior of the coefficient of discharge of cavitating nozzles. His model collapsed the experimental data and produced an interesting conceptual view of the nozzle flow. The fundamental assumptions of his model are:

1) The flow of liquid separates off the corner and contracts to an area that is a fixed fraction of the nozzle cross-sectional area.
2) The liquid experiences no head losses between the upstream stagnation state and the contraction.
3) The pressure at the contraction is equal to the vapor pressure of the fluid.

Some of the data from recent experiments [18][19] are compared with the theory explained by Nurick [17] using a plot between coefficient of discharge (C_d)and cavitation number (K). Where, cavitation number (K) given by :

$$K = \frac{(p_1 - p_v)}{(p_1 - p_2)} \qquad (2)$$

where 1 denotes upstream and 2 denotes downstream conditions. The variable p_v represents the vapor pressure.

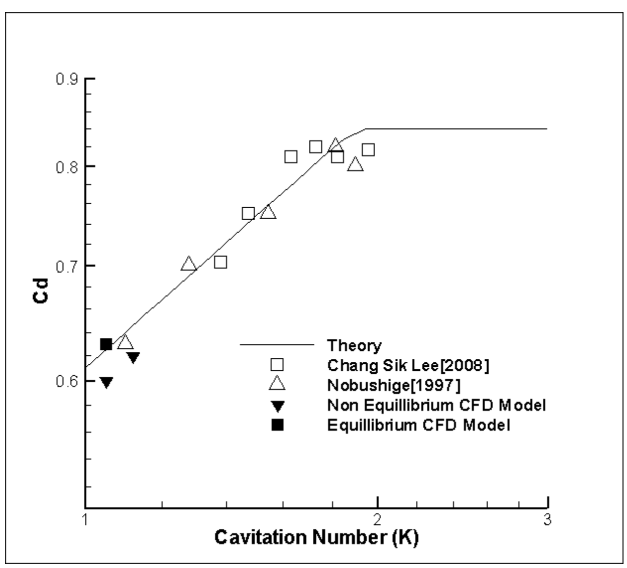

Figure 2. The graph showing experimental and CFD results for sharp-inlet nozzles from recent years compared with the theory given by Nurick [17]

Figure 2 shows that the data taken from cylindrical nozzles by Lee et al.[19] and Nobushige et al. [18] agree extremely well with the theory. For round nozzles, this further confirms Nurick's theory. (The data taken from the planar nozzle by Sou et al.[20] do not agree with the theory, probably due to a fundamental difference in the contraction behavior of planar nozzles.) Additional validation was provided by Nurick himself and by Schmidt and Corradini [21][22].

If the flow were experiencing significant thermal non-equilibrium, the third assumption in Nurick's model would not hold true. In thermal non-equilibrium, the liquid vaporizes at a pressure that is significantly below the vapor pressure of the fluid. The agreement between the experimental data and the theory indicate that, at least around the throat, the nozzles are in thermodynamic equilibrium.

Comparing simulations with and without the assumption of thermodynamic non-equilibrium provides further evidence. Figure 2 also includes a comparison between the performance of a cavitating flow solver based on thermal equilibrium and a flash-boiling flow solver based on thermal non equilibrium in simulating a cavitating nozzle flow test case. The flow conditions are based on Nobushige et al. [18] with downstream pressure of 1 atmosphere. Both the models used were based on the models developed by Schmidt [12] but were implemented for unstructured grids with complete multidimensional and parallel processing capability. The cavitation model was based on the HEM equation. The flash boiling model is based on the Homogeneous Relaxation Model (HRM) presented by Downar-Zapolski [23] and has been successfully used in the past for simulating 2D and 3D cases [24][25]. The HRM considers that the phase change process takes place over a finite time which is a function of instantaneous void fraction and pressure. Due to the Mach number limitations of the Equilibrium CFD model, the pressures were scaled upwards for the calculations, while maintaining the same value of K.

The model results indicate the flow is in thermodynamic equilibrium. The CFD calculations that assume equilibrium match the experimental results well. The non-equilibrium approach produces results that are not far removed from the equilibrium results, because the flow is so close to equilibrium. Both CFD approaches predict results consistent with Nurick's theory.

Conclusions

Based on experimental and modeling results, the assumption of thermodynamic equilibrium is sufficient to predict nozzle discharge. For future work, further comparisons can be performed to check if equilibrium assumptions will also lead to correct predictions of the length of the cavitation zone. The assumption of inertial equilibrium between the two phases is also sufficient to predict nozzle discharge and most other information required by the atomization community, because the vapor phase carries insignificant mass. The use of bubble-dynamics models, however, offers additional possibilities of predicting cavitation damage to surfaces.

Acknowledgments

We acknowledge the support of General Motors and United Technologies Research Center. We further acknowledge the support of NASA under Contract No. NNC07CB05C for this work.

References

1. Celia Soteriou, Richard Andrews, and Mark Smith, "Direct Injection Diesel Sprays and the Effect of Cavitation and Hydraulic Flip on Atomization," SAE Paper No. 950080, (1995).
2. Dennis L. Seibers, "Scaling Liquid-Phase Fuel Penetration in Diesel Sprays Based on Mixing-Limited Vaporization", SAE Paper 1999-01-0528, (1999)
3. H. Chaves, M. Knapp, A. Kubitzek, F. Obermeier, and T. Schneider, "Experimental Study of Cavitation in the Nozzle Hole of Diesel Injectors Using Transparent Nozzles," SAE Paper No. 950290, (1995).
4. Schmidt, D. P., Rutland, C. J., and Corradini, M., "A Fully Compressible Two-Dimensional Model of High Speed Cavitating Nozzle," Atomization Sprays, vol 9, pp. 255–276, (1999).
5. Kubota A., Kato H., and Yamaguchi H., "Finite Difference Analysis of Unsteady Cavitation on a Two-Dimensional Hydrofoil," Proc. of the 5th Int. Conf. on Numerical Ship Hydrodynamics, Hiroshima, pp. 667-683, (1989).
6. Chen Y. and Heister S.D., "A Numerical Treatment for Attached Cavitation," J. of Fluids Engineering, vol. 116, n. 3, pp. 613-618, (1994).
7. Arcoumanis, C., Badami, M., Flora, H., and Gavaises, M. "Cavitation in real-size multi-hole diesel injector nozzles." SAE paper 2000-01-1249, (2000).

8. Grogger, H., and Alajbegovic, A., "Calculation of the Cavitating Flow in Venturi Geometries using Two-Fluid Model," *1998 ASME Fluids Engineering Division Summer Meeting*, Washington, D.C., USA (1998).
9. Chen Y. and Heister S.D.,"Two-Phase Modeling of Cavitated Flows," Computers and Fluids vol. 24, pp. 799-809, (1995).
10. Avva R. K., Singhal A., and Gibson D.H., "An Enthalpy Based Model of Cavitation." ASME FED v. 226, pp. 63-70, (1995).
11. Delannoy Y. and Kueny J. L., "Two Phase Flow Approach in Unsteady Cavitation Modeling," Cavitation and Multiphase Flow Forum, ASME FED, vol. 98, pp. 153-158, (1990).
12. Schmidt, D.P. "Cavitation in Diesel Fuel Injector Nozzles", Ph.D. Thesis, University of Wisonsin -Madison, (1997).
13. Habchi C.,Dumont N., Simonin O., "Multidimensional simulation of cavitating flows in diesel injectors by a homogeneous mixture modeling approach", Atomization and Sprays, vol 18, pp. 129-162 (2008).
14. Kato, H., Kayano, H. and Kageyama Y., "A Consideration of Thermal Effects on Cavitation Bubble Bubble Growth", Cavitations and Multiphase flow, volume 194 FED ASME, (1994).
15. Knapp, R. T., James W. D, Hammitt F. G., Cavitation, McGraw Hill, (1970).
16. Reitz, R. D.,"A Photographic Study of Flash-Boiling Atomization". Aerosol Science and Technology. 12:561-569, (1990).
17. W. H. Nurick, "Orifice Cavitation and Its Effects on Spray Mixing," J. of Fluids Engineering, vol. 98, (1976).
18. Tamaki, Nobushige., M. Shimizu, K. Nishida, and Hiroyasu H., Effects of cavitation and internal flow on atomization of a liquid jet. Atomization and Sprays, vol 8: pp. 179-197, (1998).
19. Chang Sik Lee, Su Han Park, Hyun Kyu Suh, "Effect of cavitating flow on the flow and fuel atomization charachetristics of biodiesel and diesel fuels", American Chemical Society, Energy and Fuels, vol 22, pp. 605-613, (2008).
20. Akira Sou, Shigeo Hosokawa and Akio Tomiyama, "Effects of Cavitation in a Nozzle on Liquid Jet Atomization", International Journal of Heat and Mass Transfer, vol 50, Iss. 17-18, pp 3575-3582 (2007).
21. Schmidt D.P. and Corradini M.L., "Analytical Prediction of the exit flow of cavitating orifices", Atomization and Sprays, vol 7, pp 603-616, (1997).
22. Schmidt D.P., Rutland C.J. and Corradini M.L., "A Numerical Study of Cavitating Flow Through Various Nozzle Shapes." SAE Paper 971597, (1997).
23. Downar-Zapolski, P, Bilicki, Z. , Bolle L., and Franco J. "The Non-Equilibrium Relaxation Model for One-Dimensional Flashing Liquid Flow" IJMF, 22(3):473-483, (1996).
24. Gopalakrishnan S. and Schmidt D. P. , " Multidimensional Simulation of Flash-Boiling Fuels in Injector Nozzles" , ILASS Americas, 21st Annual Conference on Liquid Atomization and Spray Systems, Orlando, Florida, (2008).
25. Gopalakrishnan S. and Schmidt D. P. , "A Computational Study of Flashing Flow in Fuel Injector Nozzles", SAE paper SAE-08PFL-363, (2008).