International Journal of Enhanced Research in Science, Technology & Engineering
ISSN: 2319-7463, Vol. 13 Issue 5, May-2024, Impact Factor: 8.375

Study on High-Speed Flow Simulation in Fuel Injector Nozzles

Akash Suryavanshi[1], Pramod kumar[2]

[1]Research Scholar, Department of Mechanical Engineering, Rattan Institute of Technology and Management, Haryana, India
[2]Assistant Professor, Department of Mechanical Engineering, Rattan Institute of Technology and Management, Haryana, India

ABSTRACT

Atomization of fuel is essential in controlling combustion inside a direct injection engine. Controlling combustion helps in reducing emissions and boosting efficiency. Cavitation is one of the factors that significantly affect the nature of spray in a combustion chamber. Typical fuel injector nozzles are small and operate at a very high pressure, which limit the study of internal nozzle behavior. The time and length scales further limit the experimental study of a fuel injector nozzle. Simulating cavitation in a fuel injector will help in understanding the phenomenon and will assist in further development. 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 inertial effects limit phase change. 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.

INTRODUCTION

Combustion of gasoline and diesel fuel emits harmful gases and particles. The harmful gases and particles emitted from the combustion of these fuels are nitrogen oxide (NOx), carbon monoxide (CO), hydrocarbons (HC) and particulate matter (PM). Administrations all over the world, through more stringent emission standards, are keeping check on the emission parameters. Many emission standards are in use worldwide (for example, Euro V in European nations (EU) and many Asian countries). The United States government also mandated a Tier 4 standard on all on- highway diesel engines, which limit emissions to a threshold of 0.2 g/kWh NOx, 0.01 g/kWh PM and 0.14 g/kWh non-methane hydrocarbons (NHMC) [5] for 2007 and later models. In the future, introduction of a stricter standard is probable, and to meet the current and future demand, lowering of engine emissions will be required. Combustion of fuel is one of the factors affecting the emissions and developing more efficient fuel injection and engine control system can be beneficial in lowering the engine emissions.

One of the emission control strategies is to reduce the production of in-cylinder pollutants. One of the ways to reduce the production of pollutants in the cylinder is improving the combustion process. The combustion process in the cylinder can be linked with the internal nozzle flow and the atomization. In a fuel injector nozzle, atomization of fuel takes place downstream of the nozzle.

Cavitation and its role in a fuel injector nozzle

Cavitation is a phenomenon that occurs when the pressure inside a nozzle falls below the saturated vapor pressure of a liquid. This transition from liquid to vapor occurs at a constant temperature. A similar transition from liquid to vapor occurs at a constant pressure and termed as boiling (See Figure 1-2). In a fuel injector nozzle, cavitation is generally induced by an abrupt change in the geometry inside the nozzle that makes the pressure fall below the saturated pressure. Typical fuel injectors are small, and their primary role is to inject the fuel in the combustion chamber in a controlled manner. Due to a sudden contraction at the inlet of the nozzle, a boundary layer tends to separate and a re-circulation appears with a fall in pressure (see Figure 1-1). When the pressure in this region falls below the saturated pressure of the fuel, the phase transition is termed cavitation. Due to the separation at the inlet, a vena contracta is formed. This reduces the area available for the flow. The reduction in area leads to an increase in velocity at the inlet. The increase in velocity inside the fuel injector nozzle benefits the downstream atomization, which enhances the quality of the fuel/air mixture available for combustion inside the cylinder. Cavitation in a fuel injector nozzle improves the atomization through primary break up, and subsequently improves the combustion of fuel, which reduces emissions. Better atomization facilitates reduction of hydrocarbon emissions and improves the engine efficiency.

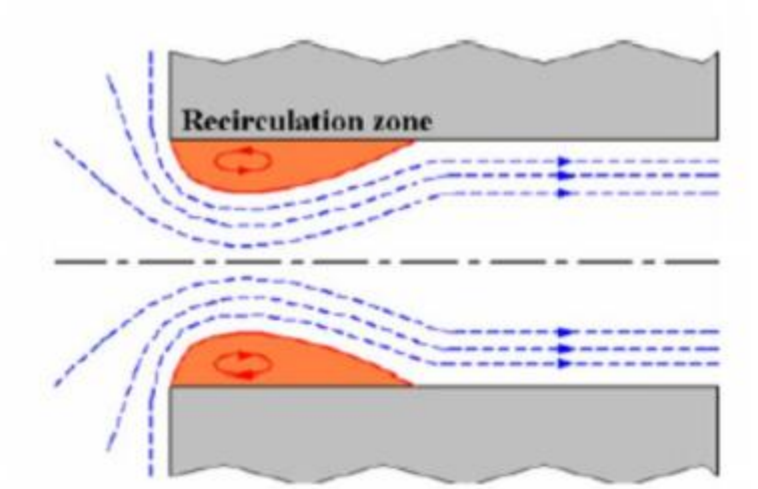

Figure 1-1 Streamlines at the inlet of a nozzle as shown by Payri

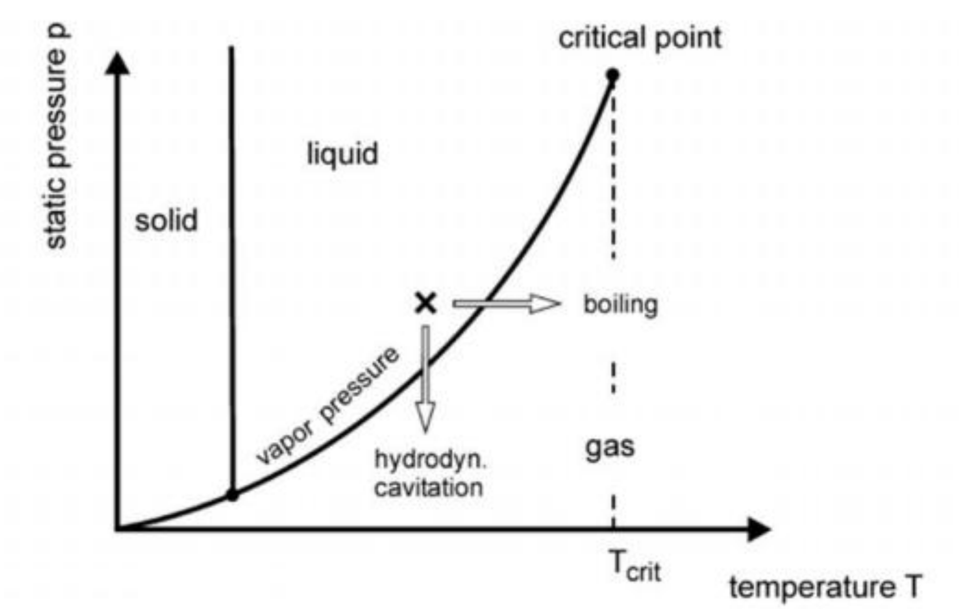

Figure 1-2 Boiling and cavitation explained in P-T diagram shown by Brenen

LITERATUREREVIEW

There are various experimental studies related to cavitation in fuel injector. In 1959, Bergwerk [13] performed one of the early observations of cavitation in fuel injector nozzle. He studied the flow in a spray hole similar in size to a real fuel injector. In his work, he related the flow through the spray hole with the influence of cavitation number, Reynolds number, upstream edge sharpness and the length to diameter ratio (L/D) and demonstrated the presence of cavitation inside the nozzle. He further mentioned flow sensitivity due to imperfections in the nozzle geometry. Decades later, Bode et al. [14] performed another study on flow through a real size transparent nozzle. Although the pressure conditions were less than real injector conditions, they observed a cavitation film appearing at the inlet corner as they increased the pressure difference. They observed bubbles collapsing outside the nozzle, as at high velocities the bubble transit time through the nozzle is smaller than the bubble collapse time. They further observed a change in cavitation inception with the change in upstream pressure. Chaves et al. [15,16] later extended this work. They used a steady flow rig with the inlet pressure varying up to 100 MPa, and the back pressure was kept at 0.1 MPa (atmospheric pressure). Parameters used by them were similar to the actual fuel injector nozzles. In their work, they showed the closeness of flow velocity and the Bernoulli velocity represented by the Equation 2.1. The Figure 2-1 demonstrates the centerline velocity plotted against the difference between upstream and downstream pressure. The dotted line represents the discharge coefficient (Cd) = 0.7 and the solid line represents Cd=1 (or Ub) [15]. The circle in the figure represents the correlation measurement value and the triangle represents the discharge measurement.

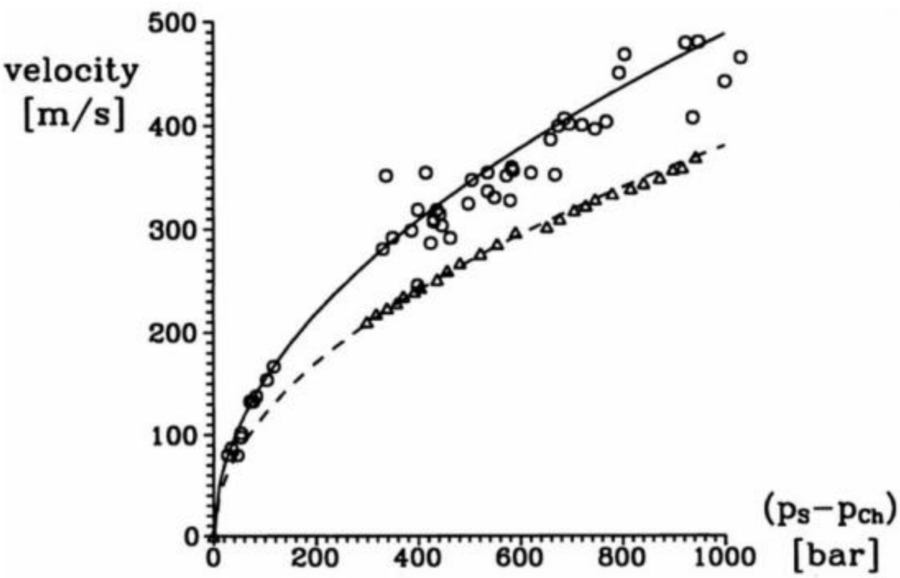

Figure 2-1 Variations in velocity with pressure difference (solid line represents C_d=1, dotted line represents C_d=0.7) observed by Chaves et al[15].

In their study, they described the phenomenon of cavities moving past the exit of the nozzle as “super cavitation”. After this condition, the discharge coefficient and the spray angle become independent of any further increase in injection pressure. Surprisingly the coefficient remains same for both short and long nozzles, although they observed a low velocity and lesser spray angle for the long nozzles [16]. The Figure 2-2 explains the increase in spray angle with the increase in pressure difference. However, after transition to the super cavitation phase, the increase in spray angle is small. In their work, they further concluded that the actual nozzle area 11 of the flow is not the geometric area. Their finding leads to a higher velocity and momentum prediction, which should be accounted for spray momentum and its effect on spray penetration.

METHODOLOGY

Governing equations the homogeneous equilibrium model used in the current study works on the basic conservation laws. A single fluid approach is used by this model, which is governed by the conservation of mass and momentum. The Equation 3.1 represents the conservation of mass and the conservation of momentum is given by the Navier- Stokes equation (Equation 3.2). Only a second coefficient of viscosity is used for the simulations.

Cavitation model

Cavitation inside a fuel injector nozzle is a very fast and transient phenomenon. The heat transfer due to bubble growth and collapse can be neglected and the flow is assumed to be in thermal equilibrium. In this model, the two-phase flow inside the nozzle is assumed homogeneous mixture of vapor and liquid. The 23 barotropic equation of state used in the model, which is used to solve the pressure, includes the compressibility of both the liquid and the vapor phase. In this work, the pure phase flow is modeled with non-linear isentropic equations, instead of the one used by Schmidt et al and other researchers, to include the non-linear effect of the flow when phase change is not dominating.

Boundary conditions

In numerical methods, different boundary condition may result in distinct solutions. Some of them may introduce non-physical influences on the domain. Arranging a correct set of boundary condition is important for physical stability inside the domain. In a fuel injector nozzle, the upstream condition consists of high-pressure fluid injecting inside the nozzle. The velocity at the far field of the upstream is close to zero. In the downstream of the nozzle, the pressure is very low and the velocity at the exit is almost 0.4 times the supersonic velocity of the fluid. Under cavitating conditions, cavitating bubbles may pass the exit boundary. Defining a boundary for such a region plays an essential role in establishing physical stability of the domain. The upstream and the downstream velocity boundaries are specified as zero gradients. A floating boundary for velocity helps the flow to stabilize. The solver is designed to simulate high-speed flows in small fuel injector nozzles.

The interaction between the solid wall and the fluid has limited viscous effects and is neglected in this simulation. A slip boundary on the wall is applied. In this approach, the upstream density is defined by a total density boundary condition. This boundary is derived from the total pressure boundary condition and uses the same equation to calculate the pressure. Since the upstream condition is in pure liquid phase, Tait-Kirkwood equation is used to calculate the corresponding density at the boundary. This boundary adjusts with the old density value at the boundary and the current local velocity. In the Equation 3.21, P0 is the total pressure set at the boundary and pos is a positive sign function which is zero, when the volumetric flux is less than 0 (i.e. the case for an inlet boundary).

CONCLUSIONS

A new solver to simulate cavitation for internal nozzle flows was constructed in OpenFOAM, an object-oriented framework that supports a variety of discretization schemes and polyhedral meshes and is parallelized using MPI. The existing cavitation solver in OpenFOAM works well with a linear compressibility model but is unstable with the more realistic compressibility models given by Wallis [32] and Chung [82] whereas this new solver performs well with the Wallis compressibility model. Besides, the new solver is capable to simulate the non-linearity in compressibility in a single-phase flow, which is not present in the existing solver. The model was validated with several experiments from the open literature. The flow simulations for the non-cavitating venturi and the sharp nozzle were consistent with the theory. The velocities obtained from the simulations of the venturi were in very close agreement with the velocity obtained from conservation of energy. The mass flow rates obtained from the simulations of the sharp-edged nozzles also correspond to the theory given by Nurick [26]. In addition, the analysis on the effective area through the nozzle during cavitation corresponds to the analytical prediction performed by Schmidt et al. [66]. The two-dimensional results displayed a shock near the exit for the submerged simulations and the Minmod TVD scheme does a good job in capturing it consistently for all the cavitation results.

The two-dimensional simulations performed with the nozzle given by Winklhofer et al. were in close agreement with the experiments. The pressure contours and cavitation probability distributions from the numerical results were in close agreement with the experiments. The comparison of the velocity profiles was lightly over predicted in simulation but the nature of the flow was closely captured. The mass flow rate from the simulations was very close to the experimental values and the incidence pattern of cavitation is accurately captured in the simulations.

SUGGESTIONS FOR FUTURE WORK

1. The meshes used in these simulations were constructed from hexahedral cells. The hexahedral mesh is difficult to generate for complex domain as compared to tetrahedral mesh but they converge faster and produce more stable results. The 85 current solver issues with high-density ratio and compressibility and instability due to poor mesh was evident. Improvement in stability with a wide range of meshes can be significant in producing efficient results.

2. The thermo physical properties for a single component fuel at equilibrium are easily available from the NIST online resources but the multicomponent fuels like diesel and gasoline are not present in open literatures. Adding thermo physical database for multicomponent fuels will enable further investigations of effects due to temperature and transport properties in internal nozzle flows.

3. Most of the second order schemes are unstable due to the presence of high gradients and shock in the flow. The Minmod scheme does well with stability and capturing shock but an implementation of ENO/WENO schemes or higher order non-oscillatory schemes in Open FOAM can be useful addition to the performance of the solver.

4. A needle motion adds significant turbulence to the incoming flow and can produce string cavitation. Implementing

needle motion to the existing study could add significant value to the solver and can be used more closely to develop fuel injectors for future diesel engines and GDI injectors

REFERENCES

[1]. Schmidt, D. P. "Cavitation in diesel fuel injector nozzles" PhD Thesis, University of Wisconsin Madison. (1997).

[2]. Schmidt, D. P., C. J. Rutland, and M. L. Corradini. "A fully compressible, two- dimensional model of small, high-speed, cavitating nozzles." Atomization and Sprays. vol. 9, issue 3, pp. 255-276. (1999).

[3]. Winklhofer, E., E. Kull, E. Kelz, A. Morozov, "Comprehensive hydraulic and flow field documentation in model throttle experiments under cavitation conditions", ILASS-Europe (2001).

[4]. Engine combustion network, http://www.sandia.gov/ecn

[5]. U.S. EPA. "Emission standards and supplemental requirements for 2007 and later model year diesel heavy-duty engines and vehicles." Paper No. 40 CFR, (2005).

[6]. Payri, R., F. J. Salvador, J. Gimeno, and J. De la Morena. "Study of cavitation phenomena based on a technique for visualizing bubbles in a liquid pressurized chamber." International Journal of Heat and Fluid Flow. vol. 30, issue 4, pp. 768-777. (2009).

[7]. Baumgarten, C. "Mixture formation in internal combustion engines." Springer. (2006).

[8]. Kashiwaya, M., T. Kosuge, K. Nakagawa, and Y. Okamoto. "The effect of atomization of fuel injectors on engine performance."SAE transactions. vol. 99, issue 3, pp. 699-705. (1990).

[9]. Montgomery, D. T., M. Chan, C. T. Chang, P. V. Farrell, and R. D. Reitz. "Effect of injector nozzle hole size and number on spray characteristics and the performance of a heavy duty D. I. diesel engine."SAE paper 962002. vol. 1205, pp. 1-18. (1996).

[10]. Kato, T., T. Koyama, K. Sasaki, K. Mori, and K. Mori. "Common rail fuel injection system for improvement of engine performance on heavy duty diesel engine." SAE Paper 980806, pp. 39-48. (1998).

[11]. Hiroyasu, H. "Spray breakup mechanism from the hole-type nozzle and its applications." Atomization and Sprays. vol. 10, issue 3-5, pp. 511. (2000) cited by [84].

[12]. Schmidt, D. P., Rakshit, S., and Neroorkar, K., "Thermal and Inertial Equilibrium in Small, High-Speed, Cavitating Nozzle Simulations," 11th Triennial International Conference on Liquid Atomization and Spray Systems, Vail, Colorado USA, July 2009.

[13]. Bergwerk, W. "Flow pattern in diesel nozzle spray holes." Proceedings of the Institution of Mechanical Engineers. vol. 173, pp. 655-660. (1959).

[14]. Bode, J., H. Chaves, F. Obermeier, and T. Schneider. "Influence of cavitation in turbulent nozzle flow on atomization and spray formation of a liquid jet." Proc. Sprays and Aerosols, ILASS Europe. (1991).

[15]. Chaves, H., M. Knapp, A. Kubitzek, and F. Obermeier. "High-speed flow measurements within an injection nozzle" SPIE. vol. 2052, pp. 265-272. (1993).