The First TSME International Conference on Mechanical Engineering
20-22 October, 2010, Ubon Ratchathani

Dynamic Characteristics of Impact Driven Jet in a Step Nozzle

Wirapan Seehanam*, Kulachate Pianthong, Wuttichai Sittiwong

Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Ubon Ratchathani, Thailand 34190

*Corresponding Author: Email wirapan_seehanam@yahoo.com, Tel. 045 353 308, Fax. 045 353 309

Abstract

High speed liquid jet can be applied and benefit in many fields such as combustion, cutting technology, and medical engineering. Liquid jet can be accelerated into high speed condition by using the methods called “Impact Driven Method” by which the liquid retained in the nozzle is impacted and then driven by a high-speed projectile. Understanding dynamic characteristics of jet generation process is essential for applying it into those technologies. So far, there are few studies in such researched areas, especially the flow inside nozzle cavity, because it is vary difficult to access in the experiment. Therefore, this study investigates the dynamic characteristics of impact driven process in a step nozzle using the Computational Fluid Dynamic (CFD) simulation. Fluid flows with transient simulation can be specified as two phase flow which consists of air and compressible diesel containing in the test chamber and step nozzle, in the initial stage, respectively. Effects of projectile velocity and mass of projectile on the characteristics of jet generation process and jet velocity are presented. Also the flow behavior due to various initial conditions is discussed in this study. It is found that the simulation shows good agreement with previously experimental results. In addition, information from this study provides the better understand on the flow phenomena of high speed liquid jet and its generation process. Moreover, the success of this study can be extended to many applications in the related fields.

***Keywords**:* Computational Fluid Dynamic (CFD), Impact Driven Method, Compressible fluid.

1. Introduction

For a few decades, much attempt has been put into researching of high-speed liquid jet for many technologies including combustion, cutting, mining, and medical engineering. Therefore, jet characteristics which are essential for such applications have been investigated.

In combustion, with higher injection pressure and the resulting higher injection velocity, the combustion efficiency of direct injected engine is increased, because the high velocity will enhance shear-induced atomization [1]. In this situation, in addition, fuel and air interactions such as jet-shock wave interaction, induced swirl, and intense shear layer have been suggested as potential ways to increase mixing during the combustion.

In the cutting technologies, the use of high speed liquid jets as a means of breaking specimens has prove to be a very promising technique. The potential for major advantage in cutting technology has been demonstrated experimentally and in practice. The advantages include considerable reduction in dust and noise generated during cutting operation, elimination of sparking, and ignition hazards, and stable equipment with lower maintenance cost [2].

For medical engineering, in drug injection, needle may be replaced with high speed liquid jet to deliver drug through skin, called "needle-free jet injection". This drug delivery benefits the improving activation, because the drug solution can become vary small particle, increasing the surface of interaction between drug and tissue. It also prevents infection in the patient and administrator by contaminated injection. In addition, diameter of the hole after injection with high speed liquid jet is very small; therefore, scar can heal up faster [3]. For drug delivery, it notes that the liquid jet velocity should be limited around 100 – 200 m/s. Moreover, nozzles with small diameter are required, usually around 0.1 mm [4]

Understanding of characteristics of high speed liquid jet and its generation process is essential to apply to those applications. Therefore, many researchers have attempted to explore the jet flow phenomena.

In 1958, F.P.Bowden *et al*. [5] showed the report of the phenomena of high speed liquid jet impact on the solid. The jets were generated by the method called "Impact Driven Method, IDM" and liquid jet at hypersonic range can be created. This method is useful to generate high speed liquid jet in the present works in the field.

With the IDM method, generally, when the liquid packaged in nozzle cavity is impacted by high speed object, shock propagations and reflections in liquid are found in H H shi and A.Matthujak's studies [6-7]. Based on this situation, Pianthong *et al.*[8] presented the one dimensional model which considered the liquid shock wave reflection for estimating the pressure of compressed liquid in step nozzle and the velocity of the high speed liquid jet emerging from the nozzle. From the model results, however, only limited parameter of the process can be predicted such as maximum injection pressure, jet velocity, and maximum compressed liquid inside nozzle, while detail of jet flow field can not be predicted and showed.

Consequently, in this study, simulation of the generation process of pulsed high speed liquid jet by using the CFD program (FLUENT) is presented. In the study, step nozzle cavity is used as geometrical model. Simulation model are validated by comparison with results from previous study's Pianthong [9], and Shi [6]. The shock waves reflection inside the nozzle cavity during jet generation process can be captured by the simulation. This clarifies how pressure buildup inside step nozzle occurs resulting in development of liquid jet and providing more understanding on high speed liquid jet phenomena and its generation process.

2. CFD modeling

2.1. Mechanism of impact driven method

The high-speed diesel jet is generated by using Bowden-Brunton method [5] as show in Fig.1. By this method, liquid retained in the nozzle is impacted by a high velocity projectile. On the impact, the high speed liquid jet forms and injects from nozzle to the test chamber.

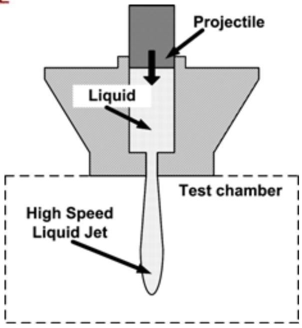


Fig. 1 Generation of supersonic liquid jet by impact driven method

2.2. CFD modeling of generation process of impact driven high speed liquid jet

Step nozzle geometry used in this study is shown in Fig 2. The geometry includes mainly two parts which are cavity and orifice tube. Variable Lc, Lo, and Dn are cavity length, orifice length, and orifice diameter respectively. Geometrical domain and grid construction are shown in Fig 3. From the mechanism of high speed jet generation shown as Fig.1, this setup can be modeled in closed domain with axis-symmetric geometry divided into nozzle cavity zone and test chamber zone. The test chamber zone, being 50 mm height and 250 mm width, was meshed with 60,000 of quadrilateral elements. This is fixed in all cases in this study, however the nozzle sac region is varied, depending on the dimension and mesh size corresponding to the nozzle cavity lengths. In this transient zone, the interval size along x-direction (dx) is fixed at 0.3 mm to provide the moving mesh for projectile motion. The mesh was densely created at the area of high shear layer and interaction between the high speed liquid jet and the quiescent air.

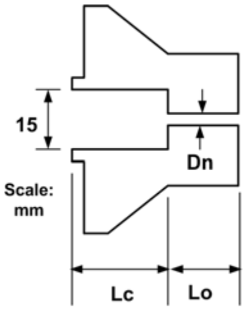


Fig. 2 Nozzle geometry

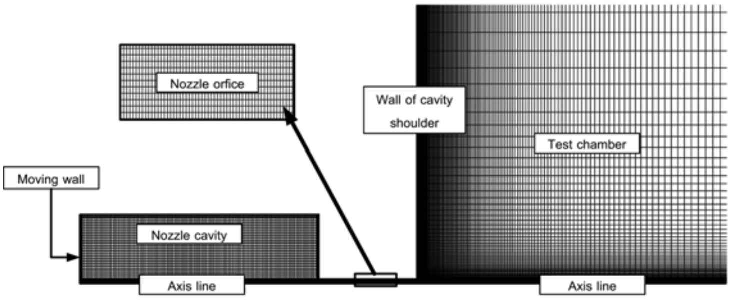


Fig. 3 Computational domain of axis-symmetric geometry of high speed liquid jet simulation

2.3. Model of projectile movement and liquid properties

The movement of the projectile in the nozzle cavity is assumed as the motion of a moving rigid wall. Therefore, the moving mesh of nozzle cavity zone was constructed. The projectile velocity equaling wall movement during jet generation process after the impact can be computed from a simple force balance on the projectile front and the liquid package in x-direction. It is assume that the force acting by the projectile, in x-direction, is simply the resistance force of compressed liquid pressure but the friction force along projectile wall is neglect. Thus, the velocity at any time t calculated by using an explicit Euler formula as

$$V_t = V_{t\text{-}\Delta t} + (F(t)/m)\Delta t \qquad (1)$$

where V is the projectile velocity, F is the driving force and m is the mass of the projectile. This formula is used to specify the motion of a moving wall (or projectile front wall) with the linear velocities at every time step (dt) through the User Define Function (UDF), provided by the software.

At the initial condition, two fluid phases were divided into liquid water phase in the nozzle cavity and air phase in the test chamber. The air density is simply specified by using ideal gas formula to cope with the compressible flow field in the simulation. Furthermore, in the nozzle cavity, it is much more complicated to specify the water as the compressible liquid. In this study, it is can be modified by using the formula including the instant liquid density (eq.(2)) and sound speed (eq.(3)) [10]. In the formula, variable P and ρ are the liquid pressure and density respectively, and the constant value B is the bulk modulus of elastic of the liquid. Subscript 0 and 1 denote the respective quantity at the initial and current time level. In addition, it seems that the density and the sound speed corresponded to liquid pressure with time dependent, significantly. Properties of diesel liquids are used in this study.

$$\rho_1 = \frac{\rho_o}{\left[1.0-\left(P_1-P_0\right)/B\right]} \qquad (2)$$

$$a_1 = \frac{1-\left(P_1-P_0\right)}{B} \times \frac{\sqrt{B}}{\rho_o} \qquad (3)$$

In addition, because of the vary high pressure gradient across two phase zones, sometimes, the pressure fluctuation can be induced by high speed liquid jet generation; consequently, some of liquid phase is evaporated to be the gas phase by cavitation process. The full cavitation models presented by Singhal *et al.*[11] and Fluent user's guide [12] are applied to specify the vapor pressure and cavitation rate in liquid and air flow. This assumption might not be accurate, but acceptable, because the liquid must evaporate to its vapor gas, instead of air. However, properties of our liquid vapor and moist air are comparable.

The CFD commercial code (FLUENT) is used as the tool to simulate the dynamics characteristics of jet generation process. The mixture model with velocity slip was used for specifying the properties of mixture within the multiphase flow. In the unsteady flow solution, the time step sized (dt) of 0.1 microseconds was set; therefore, results from each calculation can be recorded. Turbulence model is the standard k-e model with segregate solver for non-linear equations.

3. Validation of CFD simulation

This section presents the validation of dynamic characteristics of jet generation process by comparing results in this study with previous works of Painthong and Shi [7, 10]. In this study, the conditions in which projectile velocities around 300 to 700 m/s and the step nozzle are used are investigated.

Diesel liquid jet characteristics showing in term of average velocities defined as the jet penetration divided by emerging time are shown in Fig.4. These jets were driven by projectile having the velocity of 700 m/s. The average velocities calculated by the CFD method are compared with those by experimental results of Pianthong works [10]. Average overall jet velocity of both results is quite similar, about 1100 m/s, even through in the simulation the jet need more time to accelerate at the earlier stage. This indicates that, in the simulation, the penetration of high speed liquid jet might take longer time to accelerate for a few microseconds; however, in the experiment, it is not possible to capture.

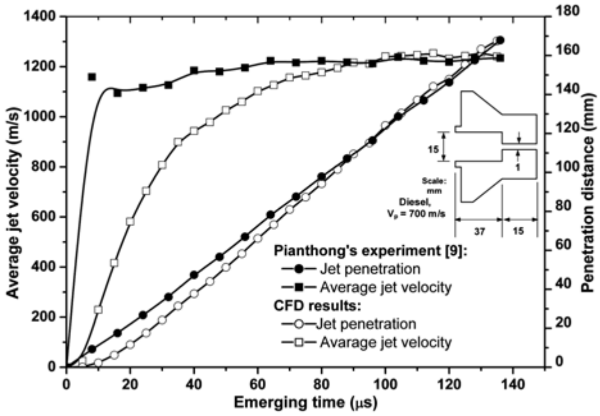


Fig. 4 Jet velocities and penetration distance

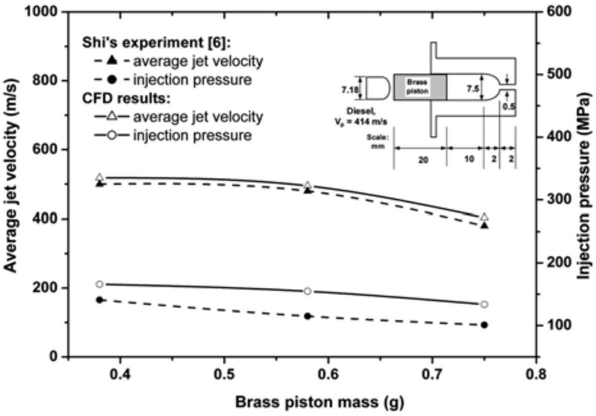


Fig. 5 Injection pressure and jet velocity with driven brass piston

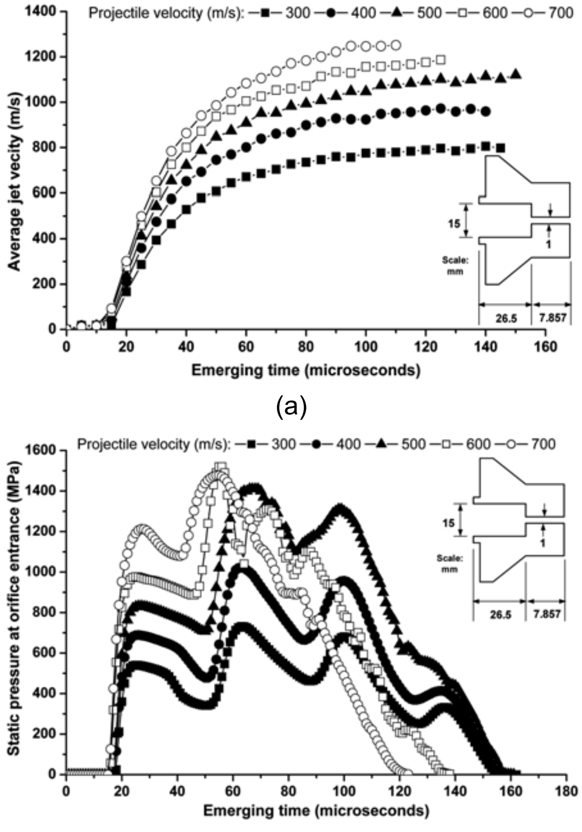
For momentum exchange method which the projectile impacts on the brass piston instead of the liquid, maximum injection pressures and average jet velocity resulting from simulation and Shi's experimental results are quite comparable as shown in Fig 5. It shows that, when the piston mass is increased, the injection pressure and jet velocity decrease. This is due to the momentum conservation which giving the slower piston movement.

4.1 Effect of projectile velocities

Because projectile velocity is the one of important parameters, many researchers have investigated the effect of the parameters on the characteristic of the injection by using experiment apparatus or mathematical model. Nevertheless, they can not thoroughly reveal how those parameters relate to the characteristics of jet injection, because in experiment it is impossible to direct measurement of jet characteristics and the parameter, especially projectile velocity and injection pressure inside. Consequently, influence of those parameters can be presented guessingly.

Therefore, this section investigates the influence of projectile velocity on dynamic characteristics of jet injection by using CFD simulation. There are projectile velocities which were ranged from 300 – 700 m/s with same nozzle geometry, and 4.2 g of projectile mass. Increasing projectile velocity can create pressure inside nozzle cavity and jet velocity to higher condition as shown in Fig 6 and Fig 7.

It is found that average jet velocity and injection pressure with the impact at high velocity rise to high values, while the number of pressure peak, being injection impulse, is independent of projectile velocity. Furthermore, the striking of projectile on cavity shoulder was found for the projectile velocity at 600 – 700 m/s. The striking and non-striking of projectile result in histories of injection pressure differently; besides, duration of jet generation process under the striking is shorter than that under the non-striking as shown in Fig. 6(b). This is because momentum of projectile was suddenly released to nozzle material during striking of projectile on the container even though it is only reduced by the liquid.



(a)

(b)

Fig. 6 Effect of projectile velocity on dynamics characteristics of high speed liquid jet: (a) velocity and (b) static pressure

The Fig 7 shows the profile of diesel jet velocity created by projectile impact velocity of 300 m/s and 500 m/s at 40, 60, and 80 μs. We observe that liquid jet impacted with high velocity projectile gives us the high jet velocity around 1400 m/s, at emerging time of 80 μs, because the maximum pressure buildup inside nozzle cavity is higher, due to lager momentum transfer. However, it is quite semblance in the jet shape. This means that the shape of liquid jet significantly relate to pressure fluctuation inside nozzle cavity.

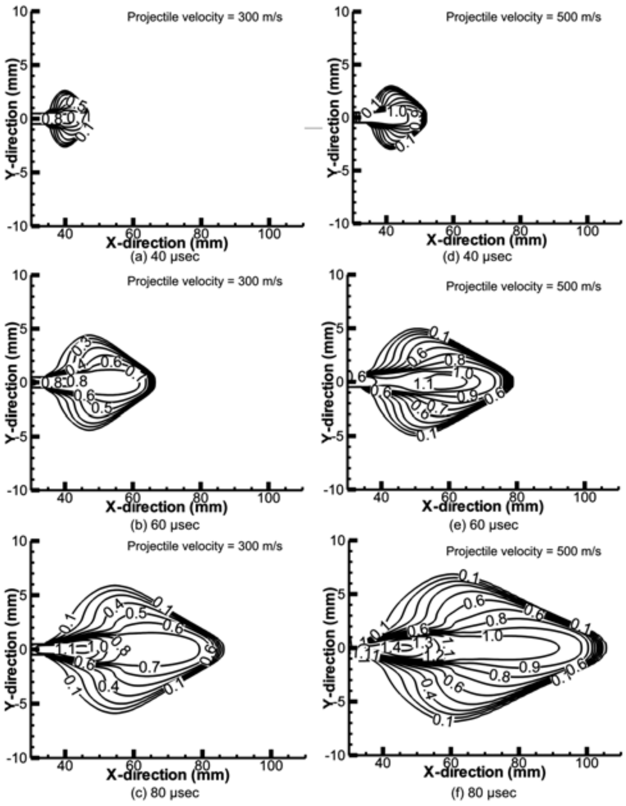
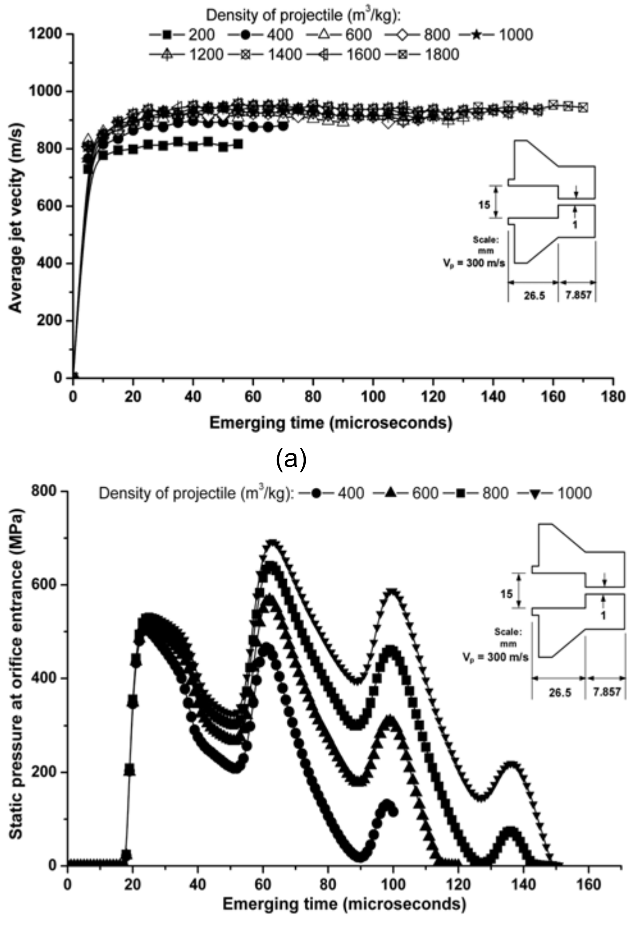


Fig. 7 Effect of projectile velocity on jet velocity profiles (km/s)

4.2 Effect of projectile density

One of important parameters at jet generation process is mass of projectile according to density of projectile at constant volume. However, previous studies have been hardly conducted such point to discussion.

In this study, it is the first time that this point is investigated. The densities being 200, 400, 600, 800, 1000, 1200, 1400, 1600, and 1800 m^3/kg and the fixed size nozzle being 4,682 cavity volumes with 7.887 Lo/Dn are used on calculation.



(a)

(b)

Fig. 8 Effect of projectile densities on dynamics characteristics of high speed liquid jet: (a) velocity and (b) static pressure

The influence performs on Fig 8 and Fig 9. From Fig 8 (a), we found that increasing projectile densities in range of lower than the liquid density does produce higher jet average velocity. However, while the density of projectile is higher than the density of the liquid, the average jet velocities are hardly varied with the projectile density. The pressure histories inside cavity are similar. However, the process – end time of jet generation is much shorter with usage of light projectile, as shown in Fig 8(b). In addition, although the first peak pressures inside nozzle value is found that it is not dependent on projectile density, the second and the third peak values are significantly varies with projectile density. It is possible for the momentum exchange at high rate with long duration after the impact of projectile.

The Fig 9 shows the profile of diesel jet velocity created by projectile densities of 300 and 500 m^3/kg at 40, 60, and 80 µs. It is found that at initial stage as 40 and 60 µs the shape of liquid jet is slightly changed with variation of projectile densities. The duration of jet generation process with light projectile is shorter than, process ending at 85 µs, such with heavy projectile, at 160 µs, resulting in difference of both jet formations of 80 µs.

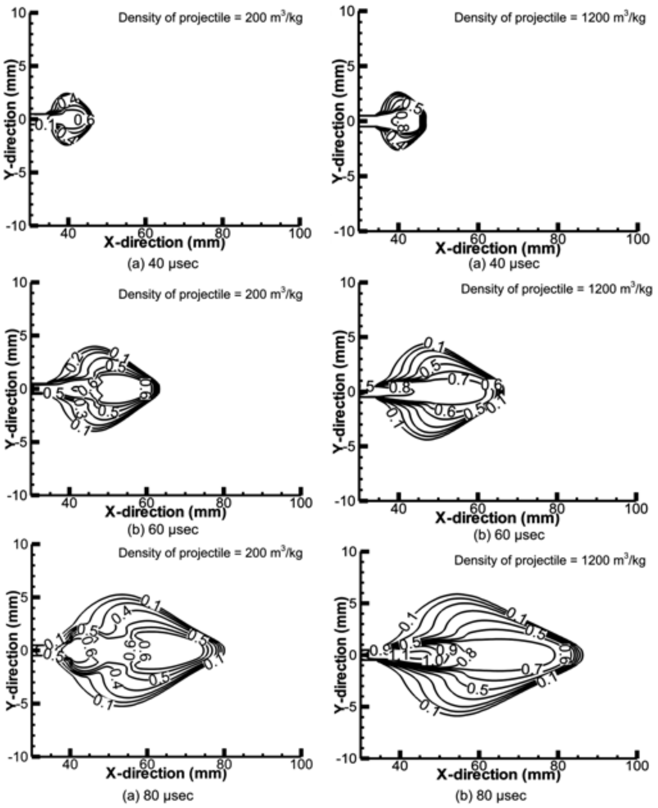


Fig. 9 Effect of projectile densities on jet velocity profiles (km/s)

5. Concluding remarks

In this study, the Computational Fluid Dynamics (CFD) technique is employed for simulation of jet generation process by IDM method within closed domain. The CFD results show good agreement to the previous experimental results. Effect of velocities and densities of projectile can be clearly investigated and described. We found that average jet velocity and injection pressure with the impact at high velocity and heavy projectile rise to high values. However, while the density of projectile is higher than the density of the liquid, the average jet velocities are hardly varied with the projectile density. Moreover, from simulation results, pressure fluctuation inside nozzle cavity considerably associate to the liquid jet formation.

6. Acknowledgement

This research is financially supported by the Thailand Research Fund (RTF), contract N0. RMU5180020, the Nation Research of Thailand (NRCT) through Ubon Ratchatani University Research Grant fiscal year 2007 and the Commission on Higher Education (CHE) of Thailand.

7. References

[1] Milton, B.E. and Pianthong, K. (2005). Pulsed supersonic fuel jet – A review of their characteristics and potential for fuel injection, *International Journal of Heat and Fluid Flow*, Vol. 25, pp.656 - 671.

[2] Shichang, Y. (1993). Dimensionless modeling and optimum design of water jet cutting system, Ph.D. Thesis, The University of Wisconsin – Madison.

[3] Giudice, E. L. and James, D.C. (2006). Needle-free vaccine delivery, *Advanced Drug Delivery Reviews* Vol.58, pp.68 – 89.

[4] Shergold, O. A. Fleck, N. A. and King, T. S. (2006). The penetration of a soft solid by a liquid jet, with application to the administration of a needle-free injection, *Journal of Biomechanics* Vol.39, pp.2593-2602.

[5] Bowden, F. P. and Brunton, J. H. (1961). The Deformation of Solids by Liquid Impact at Supersonic Speeds, *Proceedings of the Royal Society of London. Series A, Mathematical and Physical Sciences,* Vol.263, pp.433 - 450.

[6] Shi, H. H.(1994). Study of Hypersonic Liquid jets. Ph.D. Thesis, Tohoku University.

[7] Matthujak, A., Hossein, S.H.R., Takayama, K., Sun,M. and Voinovich, P. (2007). High speed jet formation by impact acceleration method, *Shock Waves*, Vol.16, pp.405 - 419.

[8] Pianthong, K., Milton, B.E. and Behnia, M. (2003). Generation and shock characteristics of unsteady pulsed supersonic liquid jets. *Journal of the International Institutes for Liquid Atomization and Spray Systems*, Vol.13, pp.425 - 620.

[9] Pianthong, K., Takayama, K., Miton, B.E. and Behnia, M. (2005). Multiple pulsed hypersonic liquid diesel fuel jets driven by projectile impact, *Shock Waves*, Vol.14, pp.73 - 82.

[10] Fluent Inc., (2005). FLUENT 6.2 UDF Manual. Fluent Inc, Lebanon.

[11] Singhal, A.K., Athavale, M.M., Li, H. and Jaing. Y. (2002). Mathematical basic and valodation of the full cavitation model, *Journal of Fluids Engineering*, Vol.124, pp.617 - 624.

[12] Fluent Inc., (2005). FLUENT 6.2 User's Guide, Fluent Inc, Lebanon.