ภาคผนวก: ผลงานวิชาการจากผลงานวิจัย

1. W. Seehanam, W. Sittiwong, **K. Pianthong**, A. Matthujak. "*Invesitgation on high speed liquid jet using computational fluid dynamics technique*". Proceedings of the 23rd Annual Conference of Mechanical Engineering Network of Thailand. CST-035222; 4-7 November, 2009. (National conference)
2. W. Seehanam, **K. Pianthong**, W. Sittiwong, "*Dynamic Characteristics of Impact Driven Jet in a Step Nozzle*" Proceedings of the First TSME International Conference on Mechanical Engineering, Ubonratchathani, Thailand, 20-22 October, 2010. (International conference)
3. W. Seehanam, **K. Pianthong** and W. Sittiwong, "*Investigation on the Characteristics of Needle - Free Injection Device*" Proceedings of the International Conference on Experimental Mechanics 2010 (ICEM 2010), Kuala Lumpur, Malaysia, 29 November - 1 December 2010. (International conference)
4. W. Seehanam, **K. Pianthong**, W. Sittiwong, B. E. Milton, K. Takayama "*CFD Investigation on Generation Process Of IDM Jets*" Proceedings of the Shock and High Strain Rate Properties of Matter (ISWI 2010), University of Cambridge, United Kingdom, September 7-10, 2010 (International conference)
5. **K. Pianthong**, W. Seehanam, W. Sittiwong and B.E. Milton "*Visualization of Injection Process in a Needle-Free Injection Device by High-Speed Video Camera and CFD*" Proceedings of the 8th Pacific Symposium on Flow Visualization and Image Processing, Moscow, Russia, 21-25, August, 2011. (International conference)
6. W. Seehanam, **K.Pianthong**, W. Sittiwong and B. E. Milton "*Injection Characteristics of Liquid Jet from a Needle Free Injection Device in the Tissue Simulant*" Proceedings of the Second TSME International Conference on Mechanical Engineering, Krabi, Thailand, 19-21 October, 2011. (International conference)
7. W. Seehanam, **K. Pianthong**, W. Sittiwong, B.E. Milton, K. Takayama "*Investigation on the generation process of impact driven high speed liquid jets using a CFD technique*" Shock Wave Journal (Under review)
8. W. Seehanam, **K. Pianthong**, W. Sittiwong, B. Milton "*Injection pressure and velocity of high speed liquid jets generated by the impact driven method*" Computers and Fluids (Under review)

Investigation on High Speed Liquid Jet using Computational Fluid Dynamics Technique

Wirapan Seehanam*, Wuttichai Sittiwong, Kulachate Pianthong, Anirut Matthujak

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

*Corresponding Author: E-mail: wirapan_seehanam@yahoo.com Tel: 0-4535-3381, Fax: 0-4535-3380

Abstract

This paper describes the simulation of the generation of high speed liquid jet injected to quiescent air including the compressible liquid dynamics in the nozzle (before injection) using computational fluid dynamics technique (FLUENT code). In this study, in the experiment, the high speed liquid jet is generated by using the impact driven method which the liquid retained in nozzle cavity is impacted by a high velocity projectile. In the numerical model, velocity of projectile driving through the nozzle was calculated with conservation equation of momentum of projectile. At initial condition, closed system domain consists of two parts which are nozzle and test chamber containing the compressible liquid and air respectively. In this study, projectile impact velocities are 300 m/s and 700 m/s. Three types of liquid jets, e.g. water and diesel were investigated. The CFD results show good agreement to the previous experimental results. In addition, simulation results proved the dynamics characteristics of multiple pulsed high speed liquid jets driven by projectile impact. From this study, it is the first reveal on the characteristics of the high speed liquid jet injected into quiescent air and liquid dynamics in the nozzle using CFD technique and will be further challenge for the study in this field.

Keywords: High-speed liquid jet, impact driven method, compressible flow, CFD

1. Introduction

There have been a number of studies of the characteristics of high speed liquid jet over a number of years. Its fundamental is essence to apply to many industrial technologies such as cutting, drilling, mining, and tunneling etc. In the combustion, moreover, the fuel sprayed to high speed liquid condition may be beneficial in improving combustion in such applications as SCRAM (supersonic combustion RAM) and direct injection. This is because the atomization and mixing are likely to be enhanced and the bow shock wave will provide significantly increased air temperatures. For medical engineering, in drug injection, needle may be replaced with high speed liquid jet to deliver drug through skin, called "needle free injection". This is drug delivery benefit which is the improving activation because drug solution can be become to be small particle, increasing the surface of interaction between drug and tissue and the preventing 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 better.[1,2] For drug delivery, it notes that the liquid jet velocity should be limited around 100 – 200 m/s, depending on design condition, which is very different from the combustion technologies. However, in both applications the high speed liquid jet can be generated by the same method called "impact driven method or impulsive method."

Impact Driven Method (IDM) technique presented by Bowden-Brunton in 1958 [3] is a method for producing high speed liquid jet. The liquid contained in cavity of station is driven by high speed projectile and accelerated to high velocity. Liquid flow behavior in liquid sac during jet generation process directly affects the characteristics of high speed liquid jet. In 2003, K. Pianthong et al.[4] presented the one dimensional model which can quite comprehensively describe the driven jet generation process during projectile traveling in the nozzle cavity. This model 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. Their model results showed good agreement to the previous experimental results. Moreover, phenomena of multiple pulsed liquid jets which were frequently presented in previous experiments can be described by their model. However, K. Pianthong et al.'s model can not be applied for other geometry of nozzle such as, mostly used, conical nozzle, because it must be calculated from two-dimensional model. A drawback in such model was confirmed by the A. Matthujak et al.'s work [5] which the second and third reflection driving pressures measured from experiment are much lower than that pressure from calculation.

Recently, numerical method such as Computational Fluid Dynamics (CFD) has been employed to investigate the high speed liquid jet characteristics. In 2003 K.Pianthong et al [6] reported the simulation of shock wave structure ahead of the jet on model of stationary solid jet shaped in steady flow field of compressible air. Then, Zakrzewski et al.[7] improved Pianthong et al's work by using the species transport equation to predict transient development of liquid jet and this improvement can describe numerically the process of interaction between the air and high speed liquid jet. Although, in previous studies, CFD results quietly agreed with experimental results, nozzle flow wasn't considered. Thus, further work from these researcher groups [8] showed simulation of shock propagating on all of material in jet generation process by using AUTODTN-2DTM software. The work shows the shock propagating in projectile, liquid sac and nozzle material but it seems that the tool can not predict transient development of high speed liquid jet.

In this study, simulation of the pulsed high speed liquid jet generation process by using the CFD program (FLUENT) is presented. In this study, step and conical nozzle cavity is the main focus, where water and diesel are used. Simulation model are validated by comparison with results from previous study's K. Pianthong [9,10] and A. Matthujak et al [5]. Static pressure at orifice entrances and dynamic pressure at orifice exit are plotted in series. Moreover profile of jet velocity is presented and discussed. This provides more understanding on high speed liquid jet phenomena and its generation process, and this information will be then useful for the future study of high speed injection and related fields.

2. Supersonic liquid jet generation by impact driven method

The high-speed fuel jet is generated by using Bowden-Brunton method [3] 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. In this paper, K.Painthong's studies [9,10] are used as main references where supersonic liquid jet was visualized by using a high speed video camera with shadowgraph optical system to describe its dynamics characteristics. In his works, the shadowgraph optical system assists in the capture of shock wave and detail of experimental apparatus was described in his studies.

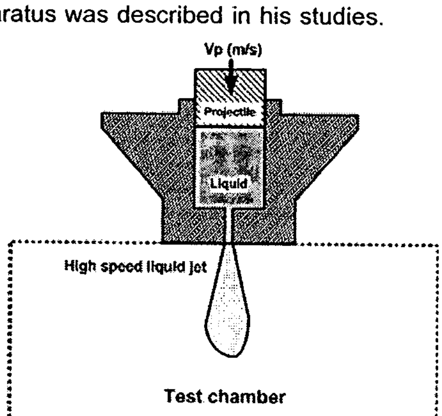


Fig. 1 Generation of supersonic liquid jet by impact driven method

3. CFD modeling of jetting formation process

3.1 Geometry model

Detail of nozzle geometry used in this study is shown in Fig 2. Mechanism of high speed jet generation is shown as Fig.1. This setup can be modeled on closed system domain with axis-symmetric geometry where was divided into nozzle cavity zone and test chamber zone (shown in Fig. 3). The 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 another zone can not do that because captured region must be changed that dimension and mesh size corresponding to the nozzle cavity lengths. In this zone, interval size along x-direction (dx) must be fixed on 0.3 mm to provide the moving mesh for projectile motion. In Fig 3, the mesh was densely created at the area of high shear flow and interaction between high speed liquid jet and quiescent air.

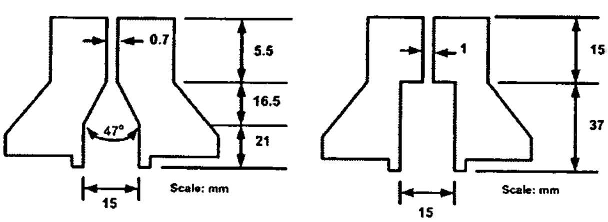


Fig. 2 Nozzle geometries

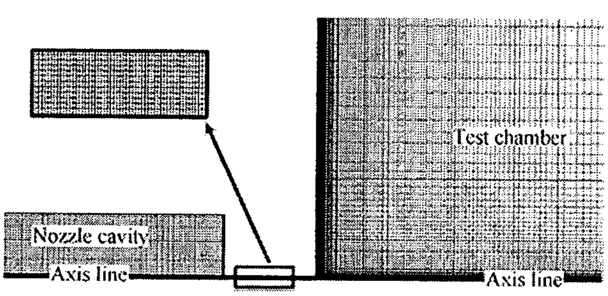


Fig. 3 Mesh construction

3.2 Projectile movement model

The movement of projectile in nozzle cavity is assumed as the motion of a moving wall. Therefore moving mesh of nozzle cavity zone was constructed. The projectile velocity during jet generation process can be computed from a simple force balance on the projectile in x-direction such that

$$\int_{t_0}^{t} dV = \int_{t_0}^{t} \left(F(t)/m \right) dt \qquad (1)$$

where V is the projectile velocity, F is the driving force and m is the mass of the projectile. The velocity at any time t calculated using an explicit Euler formula as

$$V_t = V_{t-\Delta t} + (F(t)/m)\Delta t \qquad (2)$$

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) by using User Define Function (UDF), provided by software. In this study, the mass of projectile is 4.2 g. The force acting on the projectile, in x-direction, is simply resistance force of compressed liquid pressure but the friction force along projectile wall is neglect. The projectile initial velocities being 300 m/s or 700 m/s and the atmospheric pressure are set as initial condition in the domain. Sometime, projectile might impact the nozzle shoulder, resulting from too high projectile momentum remaining. In this situation, the projectile will release such momentum into nozzle material, and its velocity is then zero before it rebounds by compressed liquid reaction force. In addition the calculation process is finished when the projectile arrive at the entry point. Because of the most different pressure across two phase zones, for some time, 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 cavitations process. Therefore, this phenomenon need to be considered in which state pressure is lower than vapor pressure of liquid. The full cavitation models presented by Ashok K. Singhal et al.[11] and [12] are applied to specify the vapor pressure and cavitation rate in liquid and air flow. This assumption might be incorrect, because the liquid must be evaporated to its vapor gas, instead of air. However, properties of our liquid vapor and air are comparable.

3.3 Liquid properties model

Under the initial condition, the fluids phase was divided into liquid phase in nozzle cavity and air phase in test chamber. The air density can be simply specified by using ideal gas formula for improving a simulation of compressible air flow. Although, in nozzle cavity, it is more complicated to specify the liquid flow to be compressible, this can be modified by using formula including the instant liquid density (eq.(3)) and sound speed (eq.(4)) [13]. In the formula, variable P and are the liquid pressure and density respectively and constant value B is the bulk modulus of elastic of the liquid, which is useful liquid property. In addition, it seems that the density and sound speed correspond to liquid pressure with time dependent. The liquids used in this study and its properties are listed in Table 1.

Table 1 Liquid properties

Liquid	Bulk modulus (GPa)	Vapor pressure (Pa)	Surface tension (N/m)
Water	2.24	3,169	0.0717
Diesel	1.6	1,378	0.0244

$$\rho_1 = \frac{\rho_o}{\left[1.0 - (P_1 - P_0)/B\right]} \qquad (3)$$

$$a_1 = \frac{1-(P_1 - P_0)}{B} \times \frac{\sqrt{B}}{\rho_o} \qquad (4)$$

3.4. Solver modeling

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.

4. Validation of CFD simulation

This section presents the validation results of dynamics characteristics of jet generation process by comparison with previous works. Water and diesel liquid jet characteristics performing in average velocity are shown in Fig.4. These jets driven by projectile with velocity of 300 m/s emerge from conical nozzle which its geometry shown in Fig.4. The average velocities calculated by CFD method are compared with such that by experimental results of K.Pianthong works [13,14]. We observe that trends of average jet velocity are slightly different. After 30 microseconds of flow time, calculated results seem that a water jet velocity is higher over that of diesel, even if there are an opposite results at over 30 microseconds, because the bulk modulus of elastic of the water is greater than that of the diesel. In addition, the diesel can form the droplet and be atomized into air easily when it was sprayed.

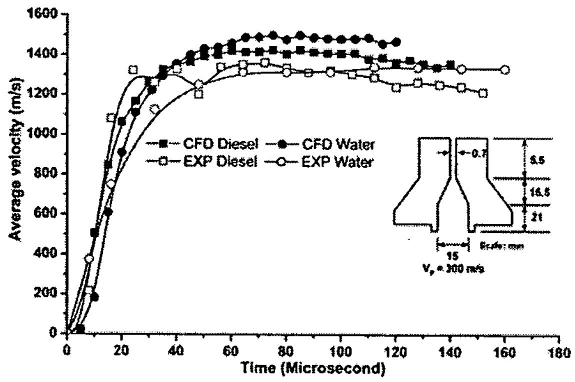


Fig .4 Average velocities

In Fig. 5, the absolute static pressure history inside the nozzle, result data from the experiment of A.Matthujak [9] and CFD simulation, are compared. In this case, the water was retained in nozzle cavity, whose geometry shown in Fig.5 and driven by projectile with velocity of 300 m/s. From Fig. 5, we found that the three peak pressures which were created by multiple water shock reflection during jet generation process. However, the pressure fluctuation corresponding to the shock waves released from nozzle container wall at initial stage can not be captured by the CFD simulation, because this situation is not considered in CFD modeling. The value of the peak pressures measured in CFD are 1.1, 0.4 and 0.3 GPa and such those in experimental results are 1.24, 0.6 and 0.27 GPa. The results from CFD are fairly similar to the experimental results, but at some stage the pressure histories from those results are more different, such as the time at 60 to 70 microsecond and 110 to 120 microseconds. Because of simple cavitation model employed in the CFD, the super cavitation process occurring during jet generation process inside nozzle cavity can not be specified by CFD model.

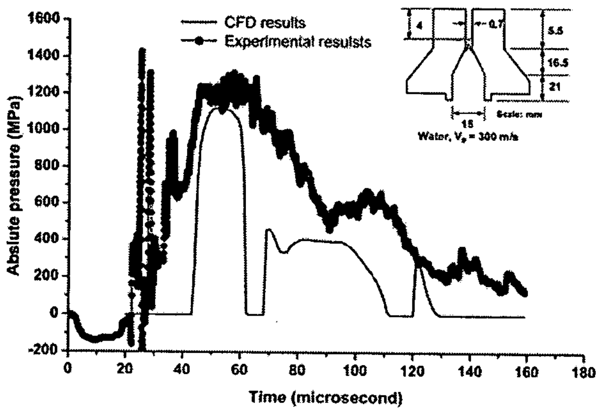


Fig .5 Pressure histories in side nozzle cavity

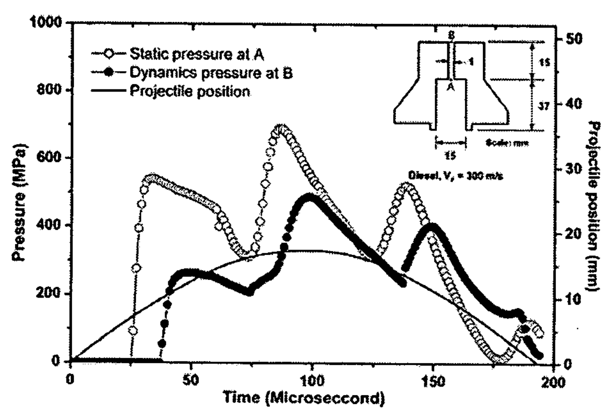
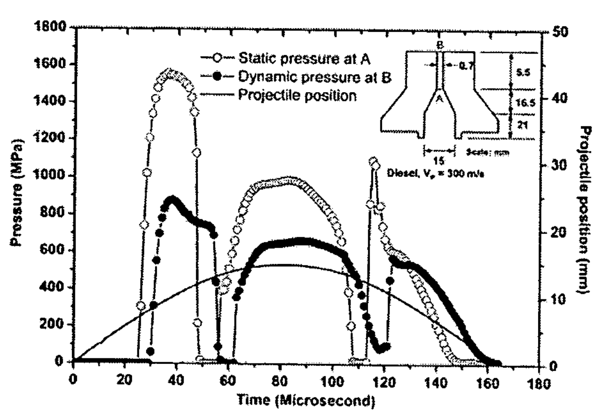


Fig. 6 Dynamics characteristics of jet generation process in (a) conical nozzle and (b) step nozzle

5. Results and discussions

This section presents the dynamics characteristics of jet generation process and high speed liquid jet obtained from CFD simulation. Position of projectile during the process, static pressure at orifice entrance and dynamics pressure at orifice exit can be plotted in series. Illustrations of profile of jet velocity are presented and described. Moreover, the effect of two nozzle geometries which are step and conical on high speed liquid jet and its generation process will be explored by using the above CFD modeling.

5.1 Relationship between static pressure and dynamic pressure



Dynamics characteristics of jet generation process which are the relationship among the position of projectile during the process, the static pressure at orifice entrance, and the dynamics pressure at orifice exit are shown in Fig. 6. We found that the static pressure is higher than the dynamic pressure. This means that liquid jet emerging from nozzle can not convert all of potential energy in nozzle cavity into kinetic energy, because while liquid jet was emerging from nozzle by driving of the static pressure, the pressure is also applied to reflectively propagate shock wave in liquid as well. Number of peak pressures in case of conical and step nozzle are three peaks similarly. However, maximum static pressure at orifice entrance occurring inside conical nozzle is higher than which inside step nozzle. In case of conical nozzle the first peak produces the highest pressure (1.5 GPa) while in the case of step nozzle the highest pressure peak occurs at second peak (0.7 GPa).

5.2 Jet velocity profile

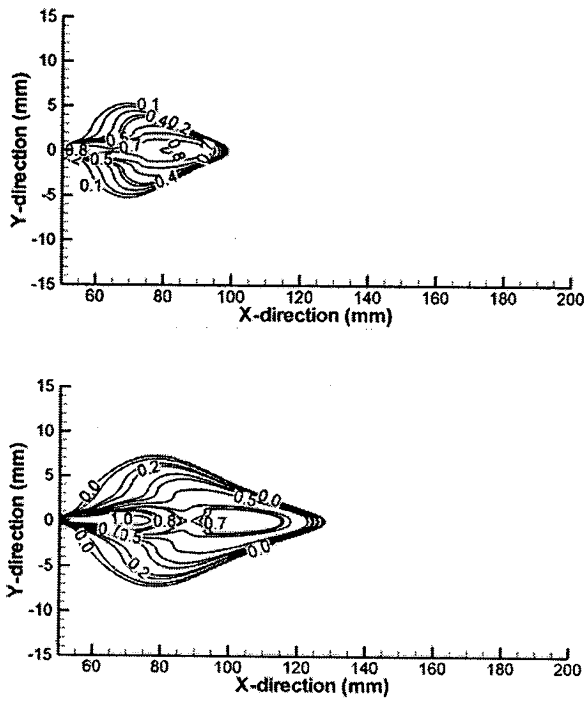


Fig. 7 Jet velocity profile of diesel emerging from step nozzle (km/s) at at (a) 90 and (c) 120 microseconds

Profile of diesel jet velocity created by projectile impact velocity of 300 m/s with step and conical nozzle is shown in Fig. 7. We observe that conical nozzle gives us the maximum velocity of 1,300 m/s, which is higher than which from step nozzle, 800 m/s, because the maximum pressure buildup inside conical nozzle is higher. Moreover, multiple diesel jet pulses occurrence can be found. The jets emerging from conical nozzle can be dispersed better than which from other nozzle, because the local velocity of the jet pulses corresponds to peak static pressure inside nozzle cavity; therefore,

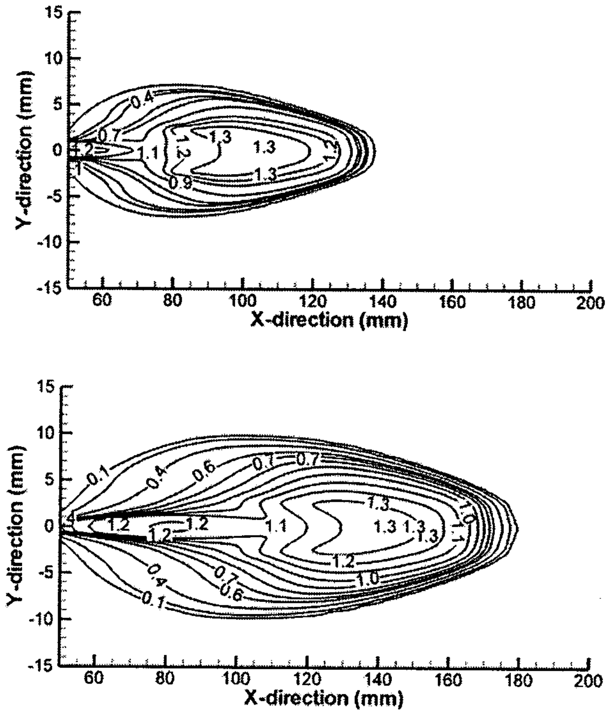


Fig. 8 Jet velocity profile of diesel emerging from conical nozzle (km/s) at (a) 90 and (c) 120 microseconds

6. Concluding remarks

In this study, high speed jets is experimentally generated by using the Impact Driven Method, from the impact of a high velocity projectile on the liquid package contained in the nozzle cavity. The Computational Fluid Dynamics (CFD) technique is employed for simulation of jet generation process by IDM method within closed system. The two fluids model consisting of liquid and air can be successfully calculated. The CFD results show good agreement to the previous experimental results. We found that in case of conical nozzle first pressure peak produces the highest pressure while in case of step nozzle the highest pressure differently occurs at second pressure peak. This is relative to local velocity and dispersion of high speed liquid jet.

7. Acknowledgement

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

8. References

[1] Giudice,E.L. and Campbell,J.D. (2006) Needle-free vaccine delivery, *Advanced Drug Delivery Reviews*, Vol. 58, 2006, pp. 68 - 89.

[2] Baxter,J.S. and Mitragotri,S. (2009) Needle-free jet injections: dependence of jet penetration and dispersion in the skin on jet power, *Journal of Controlled Release*, Vol. 97, 2009, pp. 527 - 535.

[3] Bowden,F.P. and Brunton,J.H. (1958) Damage to solids by liquid impact at supersonic speed, *Nature*, Vol. 181, 1958, pp. 873 - 875.

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

[5] 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, 2007, pp. 405 - 419.

[6] Pianthong,K., Zakrzewski,S., Behnia,M., and Milton,B.E. (2003) Characteristics of impact driven supersonic liquid jets, *Experimental Thermal and Fluid Science*, Vol. 27, 2003, pp. 589 - 598.

[7] Zakrzewski,S., Milton,B.E., Pianthong,K., and Behnia,M. (2004) Supersonic liquid fuel jets injected into quiescent air, *International Journal of Heat and Fluid Flow*, Vol. 25, 2004, pp. 833 - 840.

[8] Pianthong,K., Matthujak,A., Takayama,K., Saito,T., and Milton,B. (2006) Visualization of supersonic liquid fuel jets, *Journal of Flow Visualization and Image Processing*, Vol. 13, 2006, pp. 217 - 242.

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

[10] Pianthong,K., Matthujak,A., Takayama,K., Miton,B.E., and Behnia,M., (2008) Dynamic characteristics of pulsed supersonic fuel sprays, *Shock Waves*, Vol. 18, 2008, pp. 1 - 10.

[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, 2002, pp. 617 - 624.

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

[13] Fluent Inc. (2005) FLUENT 6.2 UDF Manual.

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.

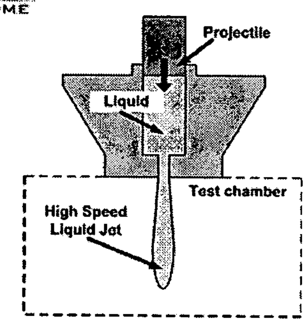


Fig. 1 Generation of supersonic liquid jet by impact driven method

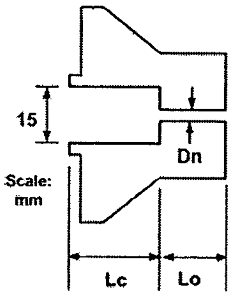


Fig. 2 Nozzle geometry

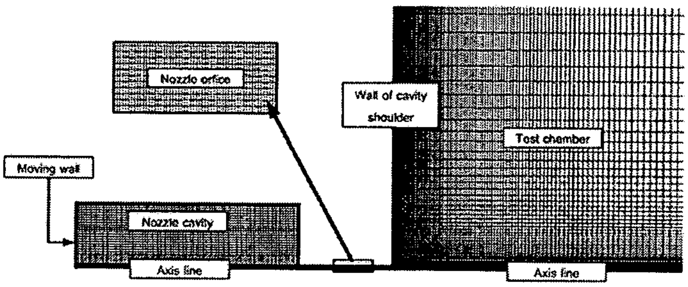


Fig. 3 Computational domain of axis-symmetric geometry of high speed liquid jet simulation

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.

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

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

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.

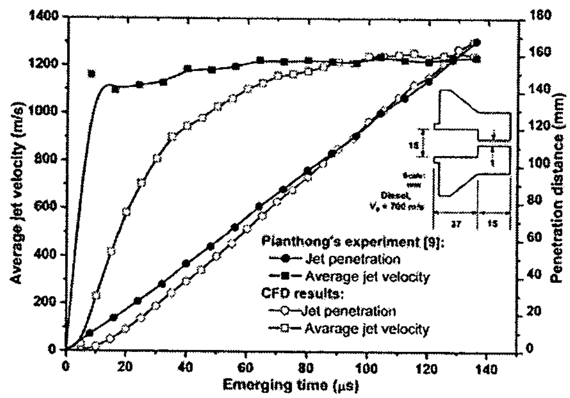


Fig. 4 Jet velocities and penetration distance

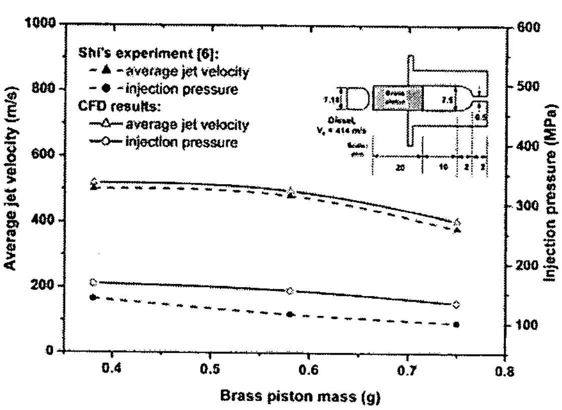


Fig. 5 Injection pressure and jet velocity with driven brass piston

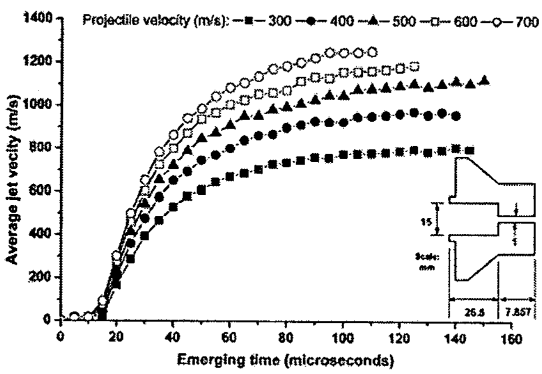
For momentum exchange method which the projectile impacts on the brass piston instead of the liquid, maximum injection pressures and average jet velocity resulting from simulation and Shi's experimental results are quite comparable as shown in Fig 5. It shows that, when the piston mass is increased,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

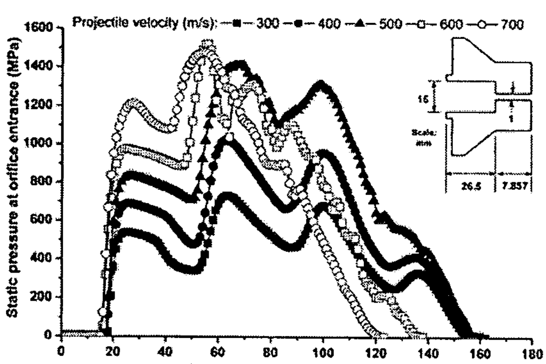
Because projectile velocity is the one of important parameters, many researchers have investigated the effect of the parameters on the characteristic of the injection by using experiment apparatus or mathematical model. Nevertheless, they can not thoroughly reveal how those parameters relate to the characteristics 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.

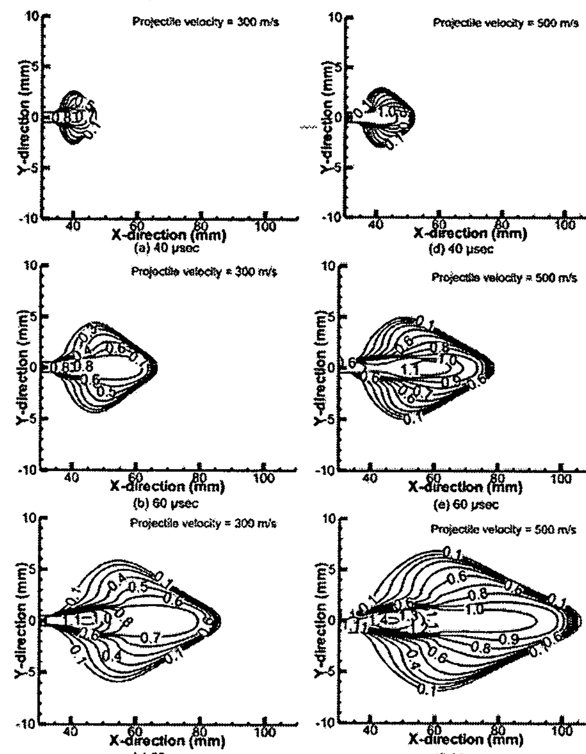


Fig. 7 Effect of projectile velocity on jet velocity profiles (km/s)

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.

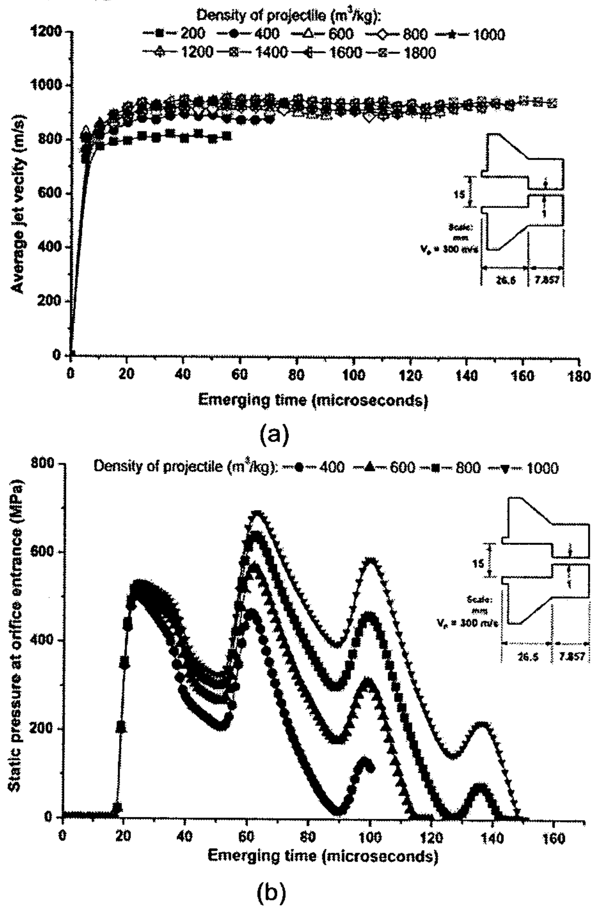


Fig. 8 Effect of projectile densities on dynamics characteristics of high speed liquid jet: (a) velocity and (b) static pressure

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.

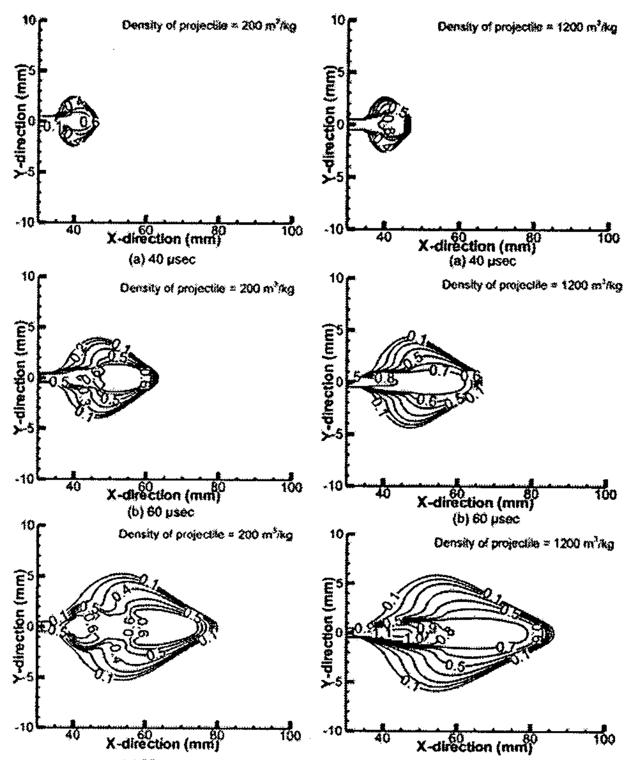


Fig. 9 Effect of projectile densities on jet velocity profiles (km/s)

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 NO. 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.

Investigation on the Characteristics of Needle - Free Injection Device

Wirapan Seehanam [a], Kulachate Pianthong [b] and Wuttichai Sittiwong [c]

Department of Mechanical Engineering, Faculty of Engineering,

Ubon Ratchathani University, Thailand

[a]wirapan_seehanam@yahoo.com, [b]k.pianthong@gmail.com, [c]sittiwong@hotmail.com

Keywords: needle-free injection, CFD, high speed video camera, impact driven method.

Abstract. In this study, the functional operation of the commercial injection device was explored, and major operating parameters, which are jet velocity, jet penetration, piston movement, and other dynamic phenomena, for the needle-free jet injection, are thoroughly investigated by using the high speed video camera. During the operation process of the needle-free injection device, the frame rate of 1,500 frames per second (fps) was applied. The water is used as working liquid and is driven through a 0.17 mm diameter orifice nozzle, into the quiescent air and the tissue stimulant being 20 % polyacrylamide gel. It is found that the velocity of the free jet in the air is around 85 m/s while the piston movement with free load is at velocity of 5 m/s. In addition, the required jet shape for the drug delivery by the injection was also observed from visualization results. Moreover, from those results, jet generation process in the device can be further simulated and investigated with CFD simulation for the better understanding of needle-free jet injection process and providing useful information in the design of needle -free jet injection apparatus.

Introduction

Jet injectors deliver liquid medication or vaccine through a nozzle orifice via a high pressure, high speed narrow stream that penetrates the skin, as shown symmetric in Fig. 1. Drug or vaccine can be delivered to intradermal, subcutaneous, or intramuscular tissue depending on operating parameters performed by the jet injector device. The devices designed to deliver the drug were first developed in the 1940's and were widely used for mass immunization campaigns from the 1950's to the 1980's [1, 2]. It is believed that jet injection devices should improve the efficacy of the drug, due to better distribution in tissues where liquid drug delivered via jet injection is dispersed more widely; in addition, site pain after injection is very small, in a range of hundred micrometers. This mechanism was confirmed by J. Baxter's studies [3-4]. An important advantage of jet injectors over other novel needle-free drug delivery methods is that parenteral delivery of drug to the same sites as those used in needle and syringe delivery may allow for use of the same vaccine formulations with the same proven efficacy [2].

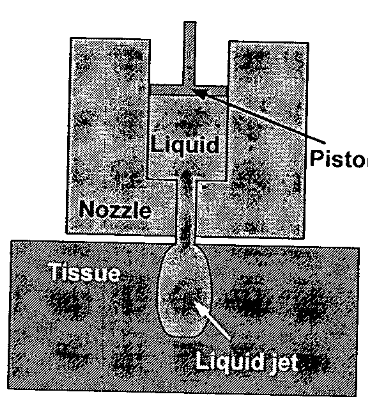


Fig. 1 Needle-free jet injection method

The device, there are many disadvantage of needle free jet injection device distributed in the market; therefore, the development have been required to improve efficiency of the jet injection method and the device. One study compared two alternative jet injector devices with standard device showed that the jet injector devices were associated with higher levels of pain and more local reactions; moreover, there is blood contamination in head of jet injection device after injection [1, 5]. This is because the device generates the liquid jet at high velocity and impact pressure resulting in blood splashed back from the patient [1, 2, 5]. For this reason, understanding on effects of the parameters on characteristics and behavior of needle-free jet injection for the completely controllable device has been essence to correctly specify the hole depth, created by the jet liquid jet, in the target tissue [2].

The devices have been concerned with injection efficiency corresponding to operating parameters which are jet penetration depth, liquid dispersion, jet velocity, volume ejected, and nozzle diameter. In works of Joy Schramm-Baxtera et al. [3, 6] mentioned that with increasing the nozzle diameter and jet velocity, the shape of liquid dispersion at the end of the hole in simulant tissue is changed and jet penetration depth is increased. Further works [4] from this research group showed that depth of the injection hole increases with ejected volume before reaching an asymptotic volume. In addition, Shergold et al. [7] explored the penetration of a soft solid by a liquid jet injection from commercial needle-free jet injection devices, and revealed the discharge characteristic. A high pressure pulse, around 20-35 MPa, during the first 1-5 ms of injection, followed by steady decay in liquid pressure was found. However, those previous studies did not explore the jet generation behavior inside the nozzle during injection process; even if, it directly affects on the characteristics of jet injection.

Therefore, in this study major operating injection parameters, which are jet velocity, jet penetration, piston movement, and other dynamic phenomena, for the needle-free jet injection, are thoroughly investigated by using high speed video camera in experiment, and this provided the preliminary data in validation of the CFD modeling of needle-free jet generation process.

Material and method

Jet production

The jets used in these experiments were produced from a commercial, spring-driven growth hormone jet injector, Cool Click [Bioject2000 Inc.] through an orifice of 170 mm in diameter. The maximum liquid volume ejected was 0.5 ml. In experiment, the jet is injected into the air and 20 % polyacrylamide gel, in which a nozzle tip attached the gel during the injection, and deionized (DI) water was used as the jet fluid. To gain a better understanding of the dynamics of the jet penetration in both medium, a high-speed video camera (Photron Fastcam SA5) was used to capture the jet flow phenomena in the medium and piston behavior during a jet ejection. During the operation process of the needle-free injection device, the frame rate of 15,000 frames per second (fps) was applied, and the major operating parameters, which are jet velocity, jet penetration, and piston movement is thoroughly investigated.

Polyacrylamide gels

20% Polyacrylamide gel were used as a model soft material which Young's modulus and hardness, Hoo, reported by Schramm-Baxter et al. [6] are 0.22 MPa and 41 Hoo, respectively. The 20% gel was created by the addition of initiators (10% ammonium persulfate (APS) and N,N,N0,N0-tetramethylethylenediamine (TEMED)) to a 40% (w/v) acrylamide solution. The acrylamide solution was mixed with DI water to create solutions possessing acrylamide concentrations in the range of 20% w/v, and the gel was polymerized by the addition of 60 ul 10% APS and 12 ul TEMED to the acrylamide solution of 6 ml.

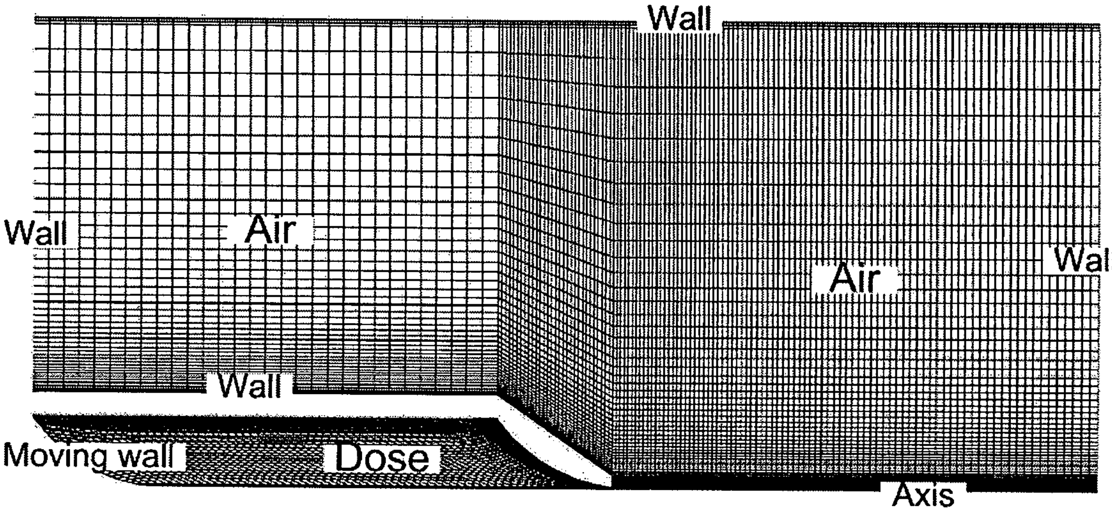
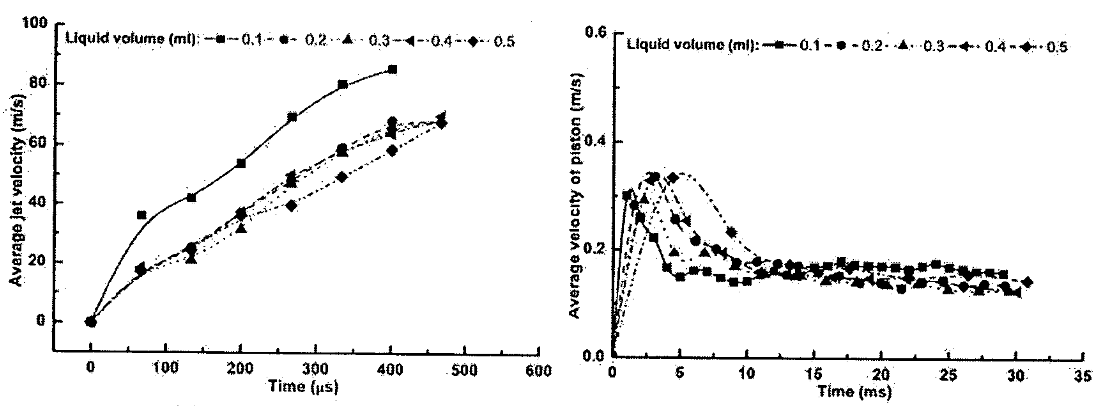


Fig. 2 CFD Modeling

CFD modeling

From the mechanism of needle-free jet injection process, this setup can be modeled in a closed system domain with axi-symmetric geometry divided into two zone: nozzle cavity zone being full of dose liquid and air zone, as shown in Fig. 2. The CFD commercial code (FLUENT) is used as the tool to simulate the dynamic characteristics of the jet generation process. In the simulation, the two-fluid model consisting of liquid and air can be calculated by using the volume of fraction (VOF) model for interaction between fluid jet and air. The air and liquid density are simply specified to be compressible fluid by using the formula of ideal gas and compressible liquid including the instant liquid density, respectively. The turbulence model is the standard k-ε model with segregate solver for the non-linear equations. The velocity of the piston movement assuming as a moving wall during the injection is computed from the resulting force from the combination of spring, pressure, and friction forces acting on the piston in x-direction. The initial spring force can be calculated from Hooke's law equation where the spring force constant tested by using Rimac Spring Tester is around 17.8 kN/m.



(a) Average jet velocity

(b) Average velocity of piston moment

Fig.3 Characteristics of the jet generation process

Results and discussion

Characteristics of the jet generation process

Dynamics characteristics of the piston and the jet injected into quiescent air, expressed as average velocities defined as its penetration distance along the medium divided by emerging time, are shown in Fig.3. It is observed that, water volume ejected is decreased, the jet velocity slightly increase, for 0.2 and 0.5 of the volume; moreover, differently higher average jet velocity is found for 0.1 ml, as shown in Fig. 3 (a). Average velocity of piston movement during jet injection process, as shown in Fig. 3 (a), is found to be steady decay over remaining 0.15 – 0. 2 m/s, before there is a high velocity pulse during the first 1-10 ms. This corresponds to the discharge characteristics, expressed as stagnation pressure, which was found by Shergold et al. [7]. In addition, duration time of the pulse is significantly extended with increasing the volume ejected, because the momentum exchange between piston and high volume liquid in a nozzle requires the long amount of the time.

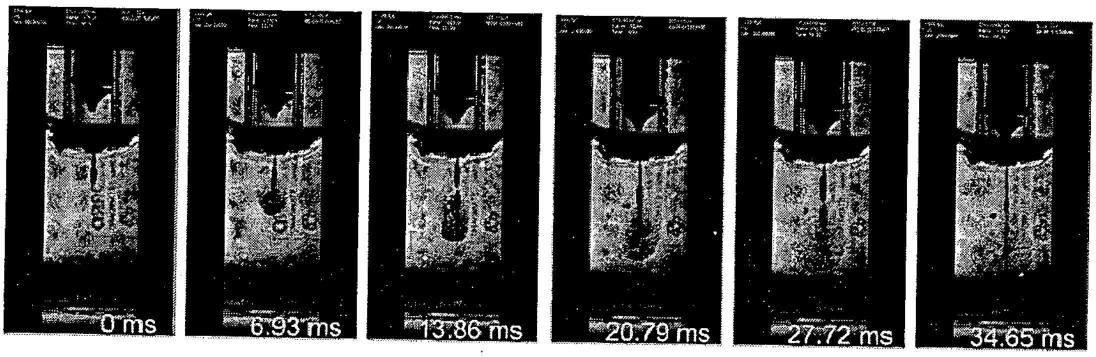


Fig 4 Dispersion of liquid in 20% polyacrylamide

Penetration behavior of jet injected into 20% polyacrylamide

Fig 3 shows the series of image of jet penetration into polyacrylamide gel. It is found that, in the first 0 - 13.86 ms, the gel is continuously penetrated by the jet liquid, especially along perpendicular, before the liquid is dispersed into the gel in all direction for 20.79 to 34.65 ms, as shown in Fig 4. That is to say jet penetration into the gels produces a hole starting at the point of jet impact to create a point source at the end of a hole, and this contribute to a circular dispersion of fluid from this point.

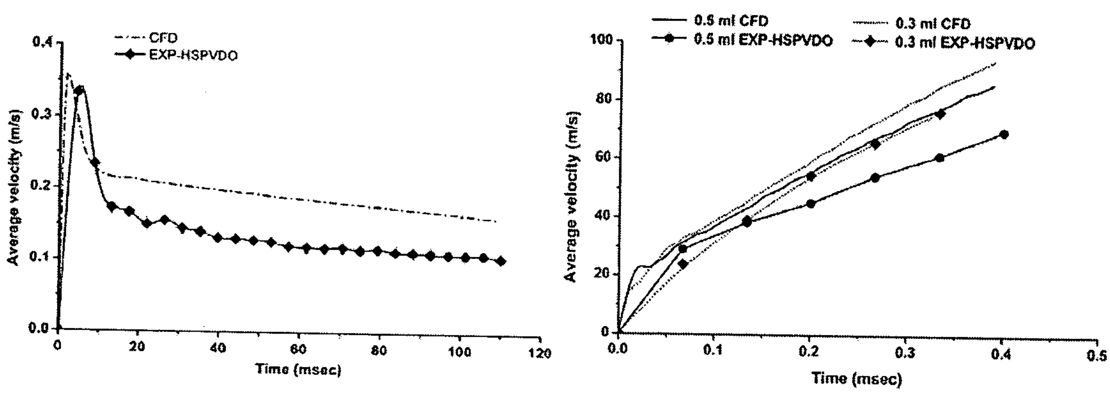


Fig 5 Average velocity

Simulation results

The average velocities of the jet ejected to the air and piston movement during jet generation calculated by the CFD method are compared with the experimental results, as shown in Fig 5. It is observed that the average velocity trends from the both method are only slightly different; although, the CFD simulation gives higher average velocities than those from experiments. In the simulation, the phenomenon of the atomization is specified by simply VOF two phase flow model, and this causes the spray atomization, corresponding to dynamics drag, occurring jet injection into the air is not fully taken into account in the CFD model. This is in accord with the results shown in Fig 6. The thin jet can be found in CFD results while cloud of liquid atomization occurring around the jet during jet injection can be captured by a high-speed video camera.

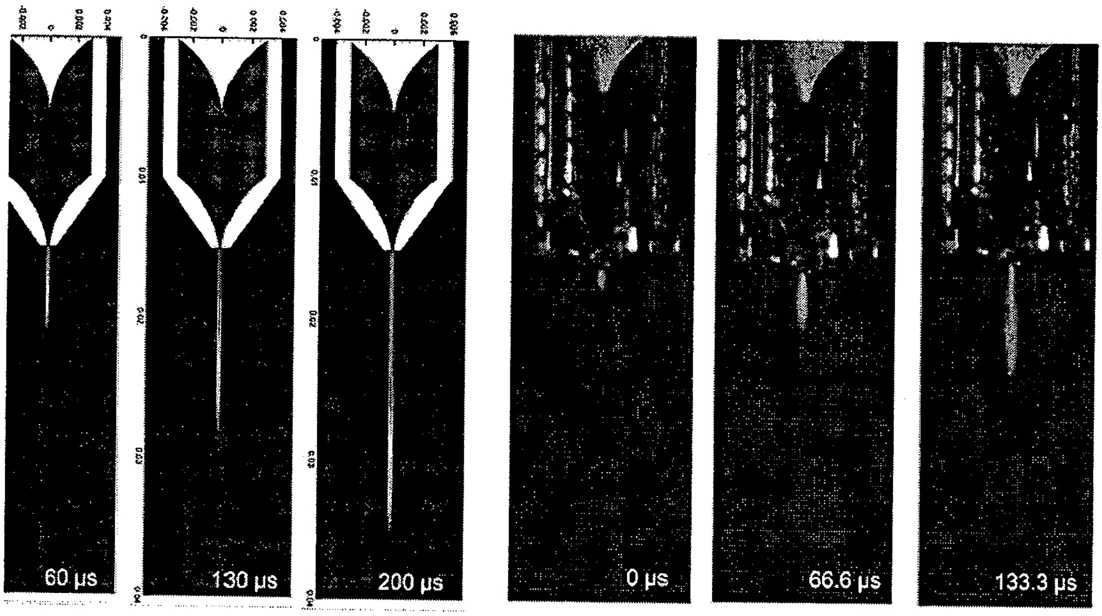


Fig 6 Visualization of the jet ejected into the air

Concluding remarks

In this study, the functional operation of the commercial injection device was explored, and major operating parameters, which are jet velocity, jet penetration, piston movement, for the needle-free jet injection, are thoroughly investigated by using the high speed video camera. The water is used as working liquid and is driven through a 0.17 mm diameter orifice nozzle, into the quiescent air and the tissue stimulant being 20 % polyacrylamide gel. It is found that water volume ejected is decreased, the jet velocity slightly increase, for 0.2 and 0.5 of the volume, and differently higher average jet velocity is found for 0.1 ml. The average velocity of piston movement during jet injection process is found to be steady decay over remaining 0.15 – 0. 2 m/s after the high velocity pulse during the first 1-10 ms. In addition, from experimental results, jet generation process in the device can be further simulated and investigated with CFD simulation for the better understanding of needle-free jet injection process. The CFD results show good agreement to the results from the experiment both quantitatively and qualitatively. The CFD modeling of the jet injection process can be applied to provide the useful information in design of needle-free jet injection device.

Acknowledgments

This work was granted by the Office of the Higher Education Commission and Thailand Research Fund (RTF), contract N0. RMU5180020, the National Research Council of Thailand (NRCT) through Ubon Ratchatani University Research Grant fiscal year 2007. Wirapan Seehanam was supported by CHE Ph.D. Scholarship

References

[1] B.E. Milton., and K. Pianthong: International Journal of Heat and Fluid Flow, Vol. 25 (2005), p. 656-671.

[2] E. L. Giudice and D.C. James: Advanced Drug Delivery Reviews 58 (2006), p. 68– 89.

[3]-24 J.Baxter and S. Mitragotri: Journal of controlled release. Vol. 97 (2004), p. 517-535

[4] -25 J.Baxter and S. Mitragotri: Journal of controlled release. Vol. 106 (2005), p. 361-373

[5]-26 L. A. Jackson, G. A. Robert, T. Chen, R. Stout, F. DeStefano , G. J. Gorse, F. K. Newman, O. Yu and B. G. Weniger: Vaccine. Vo.19 (2001), p. 4703 - 4709.

[6] Joy Schramm-Baxtera, Jeffrey Katrencikb, Samir Mitragotria: Journal of Biomechanics Vol. 37 (2004), p. 1181–1188.

[7]-28 O. A. Shergold, N. A. Fleck and T. S. King: Journal of Biomechanics Vol. 39 (2006), p. 2593-2602.

Investigation on the Generation Process of Impact Driven High Speed Liquid Jet using CFD Technique

Wirapan Seehanam, Kulachate Pianthong*, Wuttichai Sittiwong,

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

Phone: +66-4535-3309

Fax: +66-4535-3308

Email: k_pianthong@gmail.com

Abstract

High speed liquid jets have been applied to many fields of engineering, sciences, and medicines. Therefore, the investigation of its characteristics by modern and in-expensive method is beneficial to the fields. In this study, the high speed liquid jets is experimentally generated by using the momentum exchange method, called "Impact Driven Method (IDM)", by the impact of a high velocity projectile on the liquid package contained in the nozzle cavity. The shock pulse reflection within the liquid package in the nozzle resulted from the impact then causes the multiple pulsed jets. In this study, the Computational Fluid Dynamics (CFD) technique is employed to simulate the jet generation process by IDM method within a closed domain. In the simulation, two-fluid model consisting of liquid and air can be successfully calculated by using a two phase flow mixture model and a moving mesh for the projectile motion. The CFD results showed good agreement to the previous experimental study results. In addition, simulation results captured the wave propagation within liquid in the nozzle and proved the dynamic characteristics of multiple pulsed high speed liquid jets initiated by the impact driven method for the first time. This would be the breakthrough in simulation of the compressible flow of liquid and air in the supersonic ranges. It will also be vary useful fundamentals for future studies of high speed injections and related fields

Keywords: high speed liquid jets, impact driven method, Computational Fluid Dynamics (CFD), shock reflection

1. Introduction

There have been a number of studies of the characteristics of high speed liquid jet over a number of years. Its fundamental is essential to apply to many industrial technologies such as cutting, drilling, mining, and tunneling etc. In the combustion, moreover, the fuel sprayed to high speed liquid condition may be beneficial in improving combustion in such applications as SCRAM (supersonic combustion RAM) and direct injection. This is because the atomization and mixing are likely to be enhanced and the bow shock wave will provide significantly increased air temperatures. 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 [1-2]. For drug delivery, it notes that the liquid jet velocity should be limited around 100 – 200 m/s, depending on design condition, which is very different from the combustion technologies. However, in both applications the high speed liquid jet can be generated by the same method called "impact driven method or impulsive method."

Impact Driven Method (IDM) technique presented by Bowden and Brunton in 1958 [3] is a method for producing high speed liquid jet. The liquid contained in nozzle cavity is driven by high speed projectile and accelerated to high velocity. Liquid flow behavior in liquid sac during jet generation process directly affects the characteristics of high speed liquid jet. O' Keefe et al.[4] presented a development of the Bowden and Brunton technique for the production of high speed liquid jets by using a projectile with 1.77 km/s. They described the motion of water column in tapered section by applying the 1-D, unsteady equations of compressible fluid flow. Further analysis of the jet nozzle flow was mathematically presented in 1973 by Ryhming [5]. His model focused mainly on one-dimensional, incompressible flow.

Generally, when the liquid packaged in nozzle cavity is impacted by high speed object, shock propagations and reflections in liquid are always found. However, this was not considered in O' Keefe and Ryhming's study. Therefore, in 1977 Lesser [6] presented the basic mechanics of supersonic jet generation by using the theory of guides acoustic waves. His theory is the fundamental of shock propagations in liquid providing the estimation of liquid jet velocity created by IDM. Accordingly, Shi et al.'s experiment [7] showed the liquid shock reflection process in term of pressure history in nozzle cavity and described the effects of shock reflection on liquid jet characteristics. Recently, Pianthong et al.[8] presented the most popular one dimensional model which can comprehensively describe the driven jet generation process during projectile traveling in the nozzle cavity. This model 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. Their model results showed good agreement to the previous experimental results. Moreover, phenomena of multiple pulsed liquid jets which were frequently presented in previous experiments can be described by their model. However, Pianthong et al.'s model can not be applied for other nozzle geometries such conical nozzle or curved sac, because of the higher dimension effect. A drawback in such model was confirmed by the Matthujak et al.'s work [9], which the second and third shock reflection can be captured by using pressure transducer, showing in term of the pressure history. It is found that driving pressures measured from experiments are much lower than that pressure from the calculations.

Recently, numerical method such as Computational Fluid Dynamics (CFD) has been employed to investigate the high speed liquid jet characteristics. In 2003 Pianthong et al [10] reported the simulation of shock wave structure ahead of the jet with model of stationary solid jet shaped in steady flow field of compressible air. Then, Zakrzewski et al.[11] improved Pianthong et al's work by using the species transport equation to predict transient development of liquid jet and this improvement can describe numerically the process of interaction between the air and high speed liquid jet. Although, in previous studies, CFD results well agreed with experimental results, the nozzle flow characteristics wasn't considered in simulations. Thus, further work from these researcher groups [12] showed simulation of shock propagating on all of material in jet generation process by using AUTODTN-2D™ software. The work shows the shock propagating in projectile, liquid sac and nozzle material but it seems that the tool can not precisely predict transient development of high speed liquid jet.

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, nozzle cavities, including step nozzle, conical nozzle, and bell nozzle are used as geometry model to contain liquid water or diesel. Simulation model are validated by comparison with results from previous study's Pianthong [13-14], Shi [7,15], and Mutujak [9]. The shock waves reflection inside the nozzle cavity during jet generation process can be captured by the simulation. This clarifies how pressure buildup inside nozzle occurs resulting in development of liquid jet and providing more understanding on high speed liquid jet phenomena and its generation process. This information will also be very useful fundamentals for future studies of high speed injection and related fields.

2. Supersonic liquid jet generated by impact driven method

The principle of generating high speed liquid jet by using Bowden and Brunton method [3] is sketched 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. In some of previous study such as Shi's experiment [7, 15], projectile was not used to impact the liquid directly, but piston which projectile impact was used to drive the liquid into the test chamber. It is called momentum exchange method. In this paper, works of Painthong [13-14], Shi, [7, 15] and Mutthujak [9] are used as main references where supersonic liquid jet was investigated with variously experimental technique. In works of Pianthong, the shadowgraph optical system and a high speed video camera assist in the capture of shock wave in series, while the pressure history during jetting process can be captured in work of Shi and Mutthujak using pressure transducers. The detail of their apparatus was described in their study.

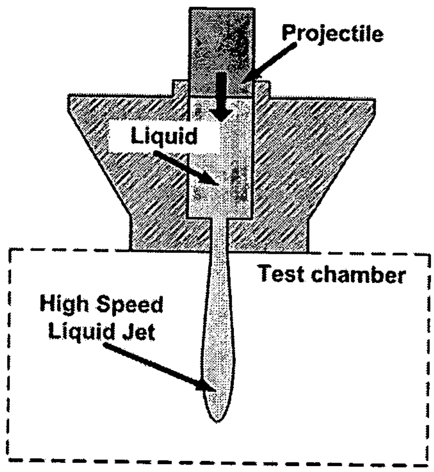


Fig. 1. Generation of high speed liquid jet by impact driven method

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

3.1 Geometry model

Details of nozzle geometries used in this study are shown in Fig 2. Two types of nozzle which are conical and step nozzle with cavity volume of 4.68 cm^3 and 6.54 cm^3, respectively, are investigated. Moreover, nozzle shapes in study of Shi [7, 15] are also used in simulation to investigate the effect of jet generation methods including; direct impact method and momentum exchange method. In this study, the commercial CFD package (FLUENT) was used. Geometrical domains of both methods are shown in Fig 3 where the bell cavity, volume of 4.20 cm^3, was used. From the mechanism of high speed jet generation shown as Fig.1, this setup can be modeled in closed system domain with axis-symmetric geometry divided into nozzle cavity zone and test chamber zone as shown in Fig. 4. 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. In Fig 4, the mesh was densely created at the area of high shear layer and interaction between the high speed liquid jet and the quiescent air.

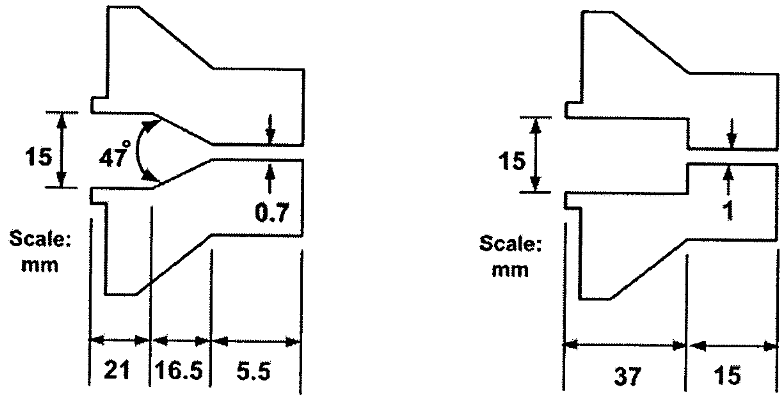


Fig. 2. Nozzle geometries (a) Conical nozzle and (b) step nozzle

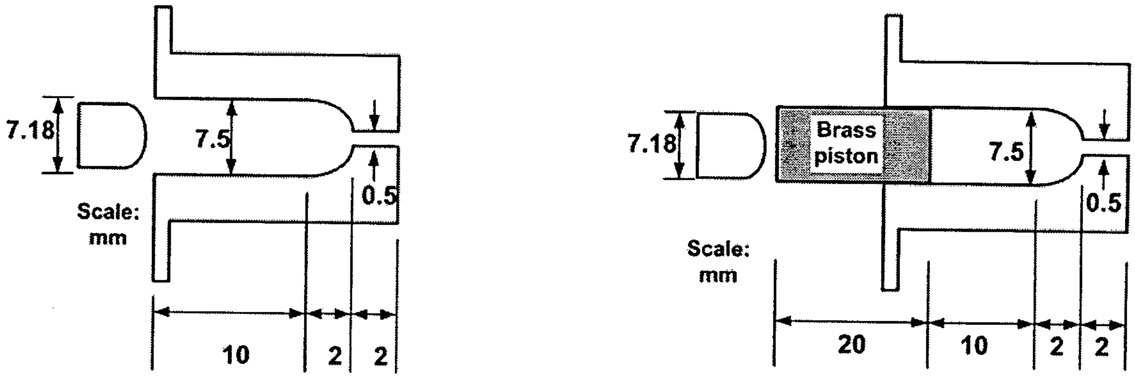


Fig. 3. Generation method (a) direct impact and (b) momentum exchange

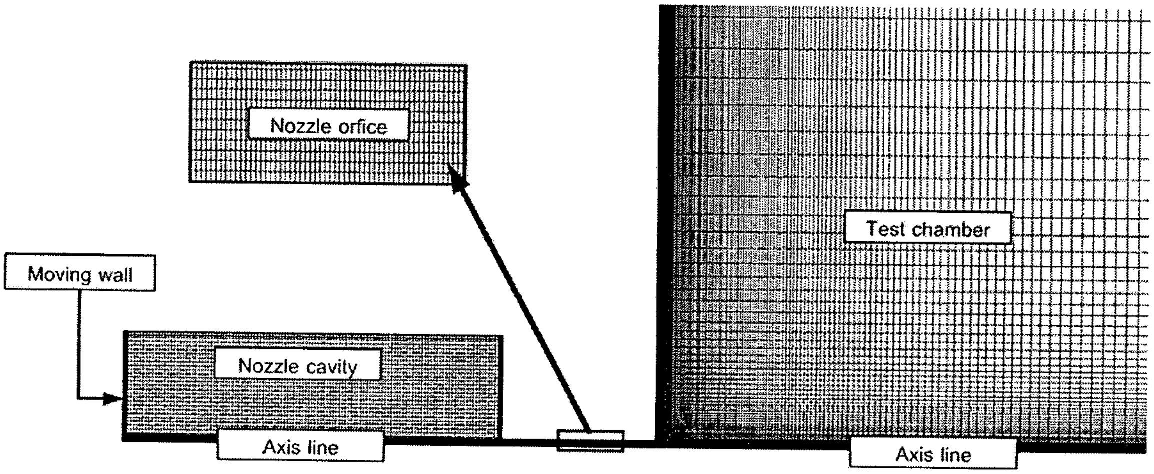


Fig. 4. Computational domain of axis-symmetric geometry of high speed liquid jet

3.2 Projectile movement model

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 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 as

$$\int_{t_0}^{t} dV = \int_{t_0}^{t} \left(F(t)/m\right) dt \qquad (1)$$

where V is the projectile velocity, F is the driving force and m is the mass of the projectile. The velocity at any time t calculated by using an explicit Euler formula as

$$V_t = V_{t-\Delta t} + (F(t)/m)\Delta t \qquad (2)$$

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. In the simulation, the mass of projectile are 4.2 g and 0.038 g which are similar to that the studies of Pianthong [13-14], Mutthujak [9], and Shi [7,15]. 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. For direct impact of projectile, such initial velocities of 300 m/s, 700 m/s, and 414 m/s following the previous study are set as initial movement of the wall, while the velocity in simulation of momentum exchange method can be computed from formula of conservation of momentum in the system of the projectile and brass piston. The atmospheric pressure and ambient temperature are set as initial condition in the domain. Sometime, projectile might impact the nozzle trap, resulting from too high projectile momentum remaining. In this situation, the projectile will release such momentum into nozzle material, and its velocity is then zero before it rebounds by compressed liquid reaction force. In addition the calculation process is finished when the projectile arrive at the entry point. Because of the most different pressure 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. Therefore, this phenomenon needs to be considered when the local pressure is lower than vapor pressure of liquid. The full cavitation models presented by Singhal et al.[16] and Fluent user's guide [17] 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.

3.3 Liquid properties model

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.(3)) and sound speed (eq.(4)) [18]. 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. Liquids used in this study and their properties are listed in Table 1.

$$\rho_1 = \frac{\rho_o}{\left[1.0-(P_1-P_0)/B\right]} \tag{3}$$

$$a_1 = \frac{1-(P_1-P_0)}{B} \times \frac{\sqrt{B}}{\rho_o} \tag{4}$$

Table. 1. Properties of water and diesel used in this study

Liquid	Bulk modulus (GPa)	Vapor pressure (Pa)	Molecular weight (g)	Specific heat (j/(kg.K)	Surface tension coefficient (N/m)
Water	2.24	3,169	18	4,182	0.0717
Diesel	1.6	1,378	170	1,850	0.0244

3.4 Solver modeling

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.

4. 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. Water and diesel liquid jet characteristics showing in term of average velocities defined as the jet penetration divided by emerging time are shown in Fig.5. These jets driven by projectile having the velocity of 300 m/s, emerge from conical nozzle which its geometry is shown in Fig.2 (a). The average velocities calculated by the CFD method are compared with those by experimental results of Pianthong works [13-4]. We observe that trends of average jet velocity are just slightly different. At first 30 microsecond, the experimental and CFD results are very closed, however, at the after stage, the CFD simulation gives higher average velocities than those from experiments. Also, calculated results show that the water jet velocity is higher over that of diesel, even if there are an opposite results at over 30 microseconds, because the bulk modulus of elastic of the water is greater than that of the diesel. For a large the modulus, the fluid volume can be laboriously changed and the built-up of high pressure is created when changing the external pressure working upon the fluid, In addition, the diesel jet may form the droplet cloud and be atomized into air easier when it start to breakup, because of low its surface tension.

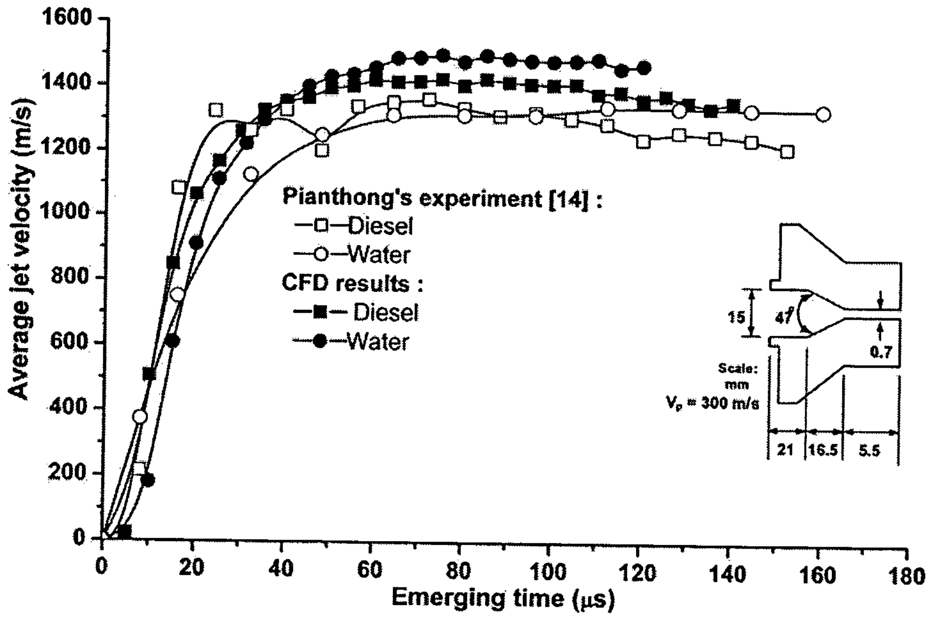


Fig .5. Average jet velocities

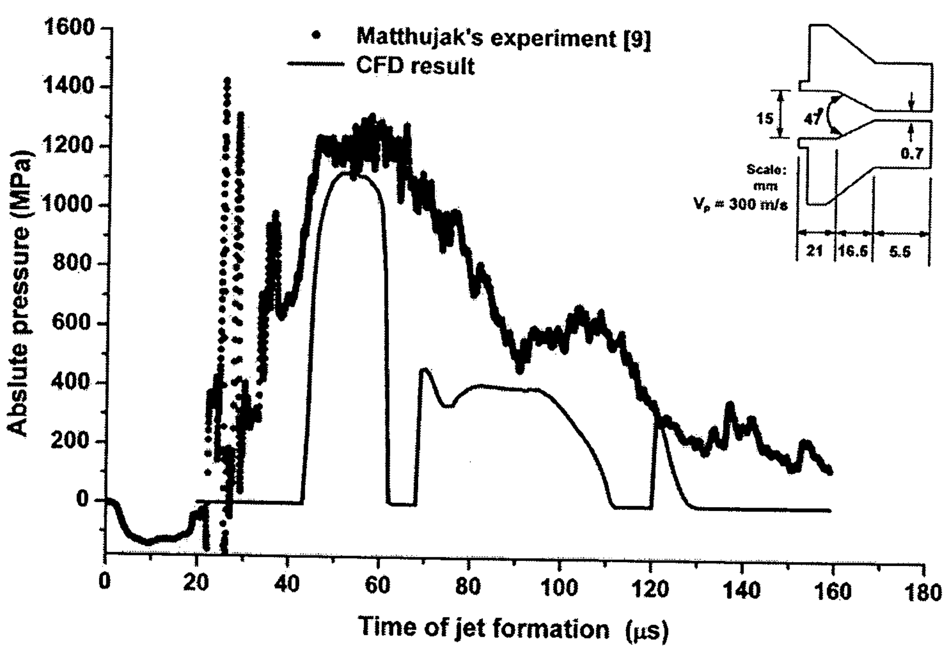


Fig. 6. Pressure history inside the nozzle cavity

In Fig. 6, the absolute static pressure histories inside the nozzle from the experiment of Matthujak [9] and CFD simulation are compared. In this case, the water was retained in the nozzle cavity whose geometry shown in Fig.2(a) and driven by the projectile with velocity of 300 m/s. From Fig. 6, we found that there were three peak pressures which were created by multiple water shock reflection during jet generation process. However, the pressure fluctuation corresponding to the shock waves released from nozzle container wall at initial stage (20 microseconds) can not be captured by the CFD simulation, because this situation is not considered in CFD modeling. The value of the peak pressures obtained from the CFD are 1.1, 0.4 and 0.3 GPa while those values from the experiment are 1.24, 0.6 and 0.27 GPa. The results from CFD are fairly similar to the experimental results, only at some stage the peak pressure from those results occurs at different period (i.e. the peak pressure of 2nd and 3rd pulse). The experiment gives slightly delay, around 20 – 30 microseconds, corresponding to CFD. Because of simple cavitation model employed in the CFD, the actual super cavitation process occurring during jet generation process inside nozzle cavity was not really taken in to account in the CFD model.

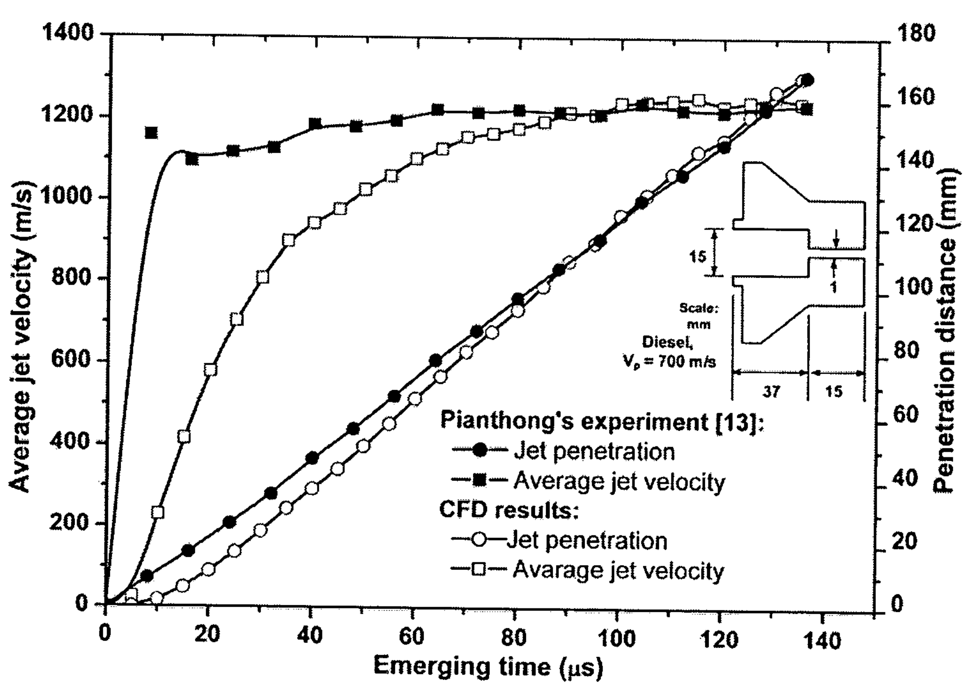


Fig .7. Jet velocity and penetration distance

Fig. 7 shows the characteristics of high speed diesel jet in term of velocity and the penetration distance emerging from step nozzle which its geometry is shown in Fig.2 (b). This jet was generated by the impact of the projectile with velocity of 700 m/s. Experimental results of Painthong's study, in 2003 [13], are compared with the simulation results. 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 capture. At 0 to 60 microseconds, average jet velocities from experiment are higher than that from calculation. This is because, in the experiment, shock propagation in container material resulting from projectile impact may be released into the liquid at the earlier stage, while this situation dose not consider in the simulation.

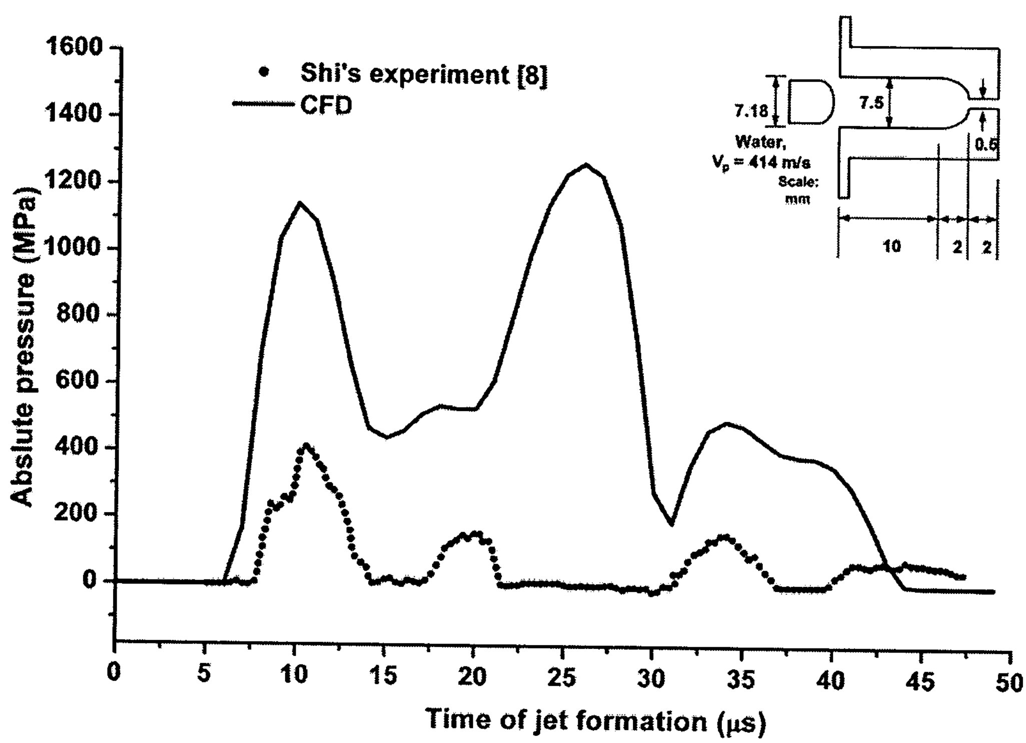


Fig. 8 Pressure history from direct impact of projectile

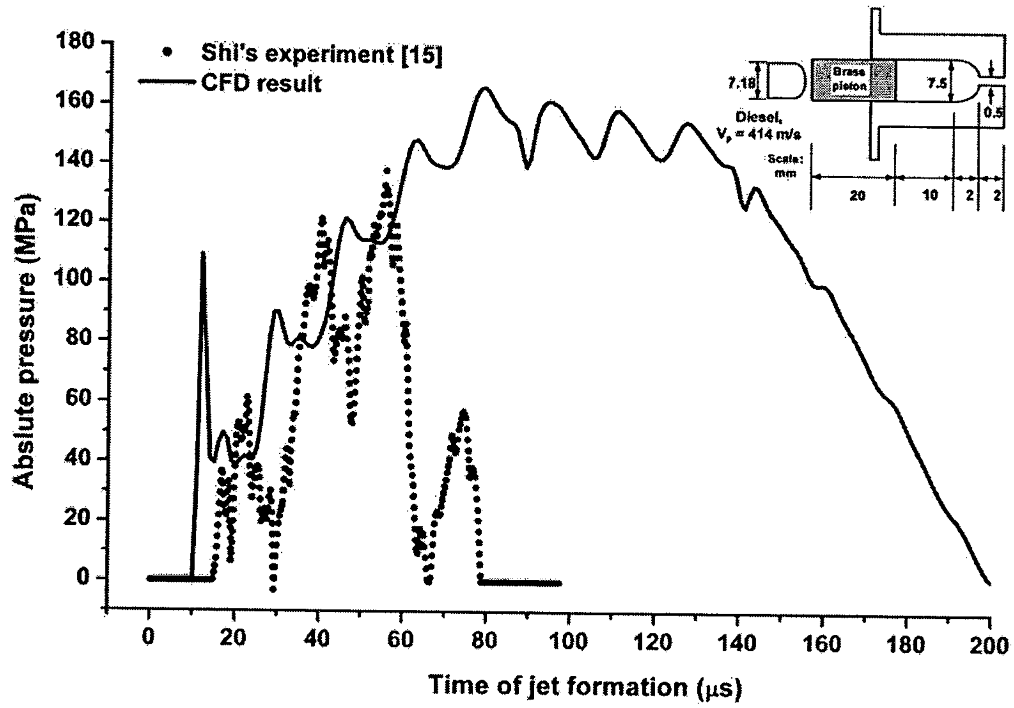


Fig. 9 Pressure history with driven brass piston

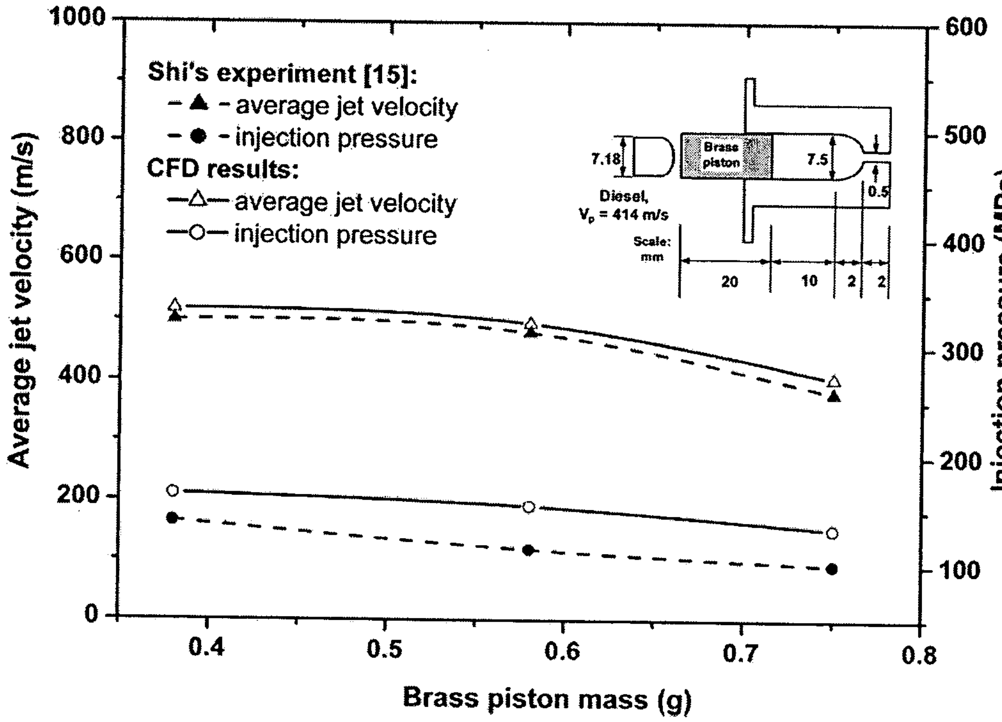


Fig. 10 Injection pressure and jet velocity with driven brass piston

Moreover, the CFD results significantly agree with experimental results from Shi's studies [8, 15] in which two impact methods including direct impact method and momentum exchange method are used for generating high speed liquid jet. Retained liquid was impacted by the projectile, mass of 0.384 g and 414 m/s of impact velocity. Pressure inside nozzle cavity and average jet velocity which were achieved from CFD simulation and their experiment are shown in Fig 8-10.

For direct impact method, maximum driving pressure from calculation and experiment is significantly different. However, the trend of pressure history agrees quite well corresponding to pressure peaks during jet generation process as shown in Fig 8. Because of CFD model which is closed system with all boundary condition set as walls, leak of liquid through tolerance between the nozzle hole and the projectile can not be specified; therefore, higher injection pressure may be given from calculated results. In addition, large tolerance between projectile and nozzle cavity was used in Shi's experiment, around 0.3 mm (7.18 mm and 7.8 mm diameter of projectile and nozzle cavity respectively) and efficiency of the impact of experimental apparatus also deal with uncertainty results from experiment. However, this slightly affects on shock reflection of liquid during the process. This is confirmed in the comparison between both methods as shown in Fig 8.

For momentum exchange method which the projectile impacts on the brass piston instead of the liquid, maximum injection pressures resulting from simulation and Shi's experiment are quite comparable as shown in Fig 9: the pressure given from both case are around 145 MPa. Actually physical domain of this method approaches nearby the closed domain in the simulation. Similarly, the CFD shows good agreement with experimental results as shown in Fig 10 which average jet velocity and maximum injection pressure relate to the piston mass. Fig 10 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. However, duration of jet generation process from CFD is twice longer than that from the experiment. This is because of the simulation assumptions setting the projectile always attach with the piston and move together along nozzle cavity. Differently, in the experiment, the projectile may bounce off the piston during the process giving the duration of the setting process is significantly short. Moreover, in the experimental condition the fiction force will occurs between the piston and the nozzle cavity wall as well.

5. Dynamic characteristics of jet generation process

This section presents the dynamics characteristics of jet generation process resulting from CFD simulation. Illustrations of pressure profiles inside nozzle cavity is presented and described. Behavior of liquid shock reflection in nozzle cavity has most significant effect on high speed liquid jet characteristics. Therefore, this paper shows the instant pressure profile inside nozzle, as shown in Fig. 11. and 12, to observe the profile in series. The instant static pressure corresponding to shock wave in the diesel column containing in step and conical nozzle cavities having volume of 4,682 and 420 mm^3 is generated by impact of projectile with velocity of 300 and 414 m/s respectively.

In step nozzle condition, in Fig 11, it is shown that pressure buildup near the nozzle shoulder occurs for three times corresponding to number of pulse of jet. At the earlier stage, Fig 11(a), the static pressure around 0.3 GPa near the projectile front which moves forward on x-direction is higher than the pressure further away. Pressure is built slightly up first near the nozzle shoulder at 30 microseconds of flow time. When the pressure can be adequately accumulated over maximum pressure in earlier stage, reflected shock occurs and then travels backward to projectile face, while the high speed liquid jet forms. After the reflection shock wave is near the projectile face, it reflect back again by buildup pressure. Therefore the pressure was added up over shock reflection process. This is clear explanation on the formation of the multiple pulsed of the high speed liquid jet.

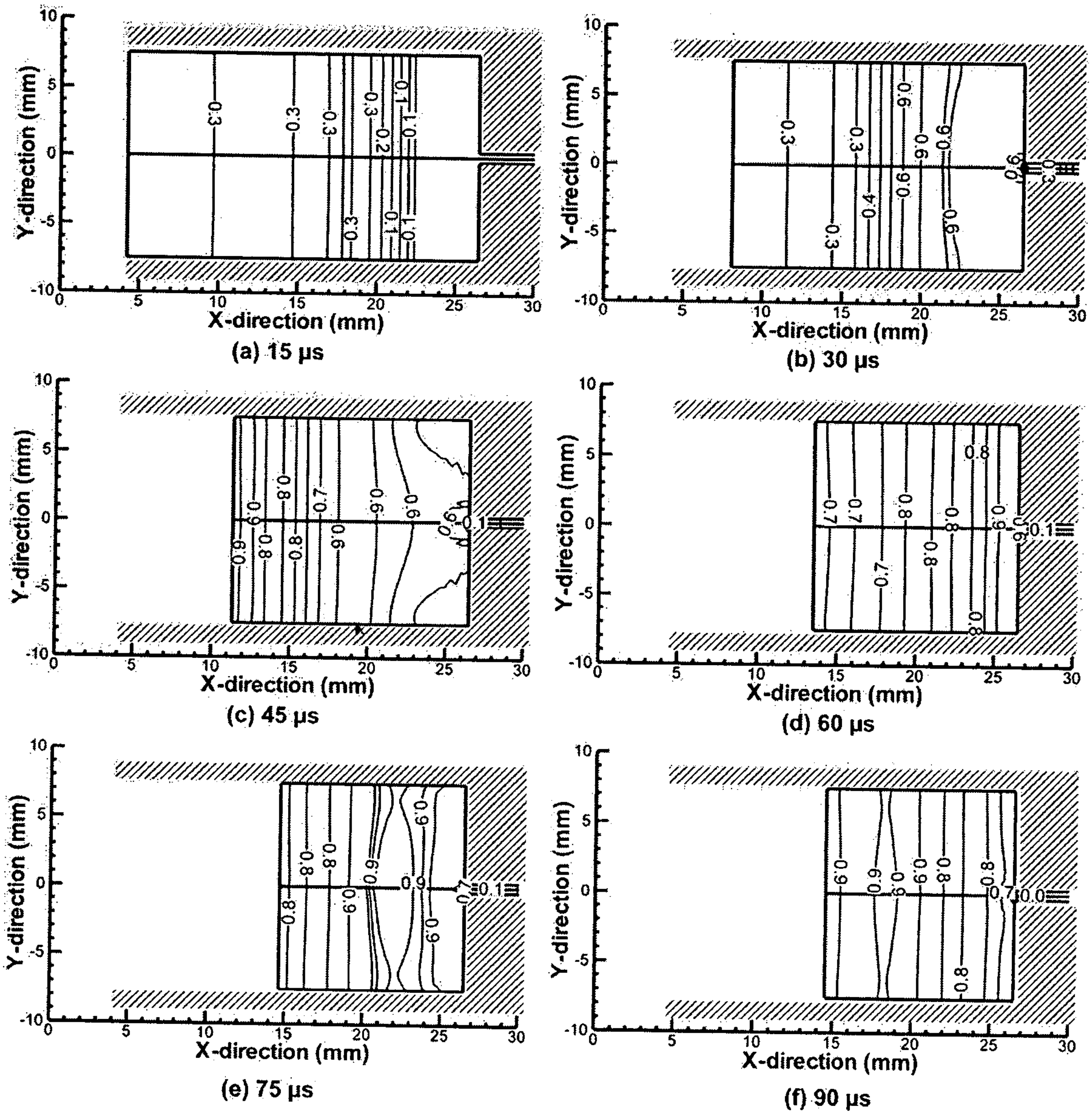


Fig. 11 Pressure profile (GPa) of diesel inside step nozzle

Pressure profiles inside step and conical nozzle cavity is not similar in pressure buildup process near the orifice entrance. Therefore, the value of driving peak pressure in both cavities is significantly different. From Fig 12, at earlier stage (Fig 12 (a)), the high speed liquid flow resembles the converging flow through the nozzle, the conical nozzle can create the highest pressure at the first process of pressure buildup. Later, the pressure can be greatly reduced by releasing the liquid into the air (as shown in Fig 12c). This phenomena is different to that in the step nozzle cavity resembling the cylinder flow as shown in Fig 11. The maximum pressure inside the cavity is at second process of pressure buildup near the orifice entrance because of reduction of pressure at the first process where liquid is scantly released, remaining high pressure to be accumulated again. However we found similarity a third such in both case during rebound of projectile.

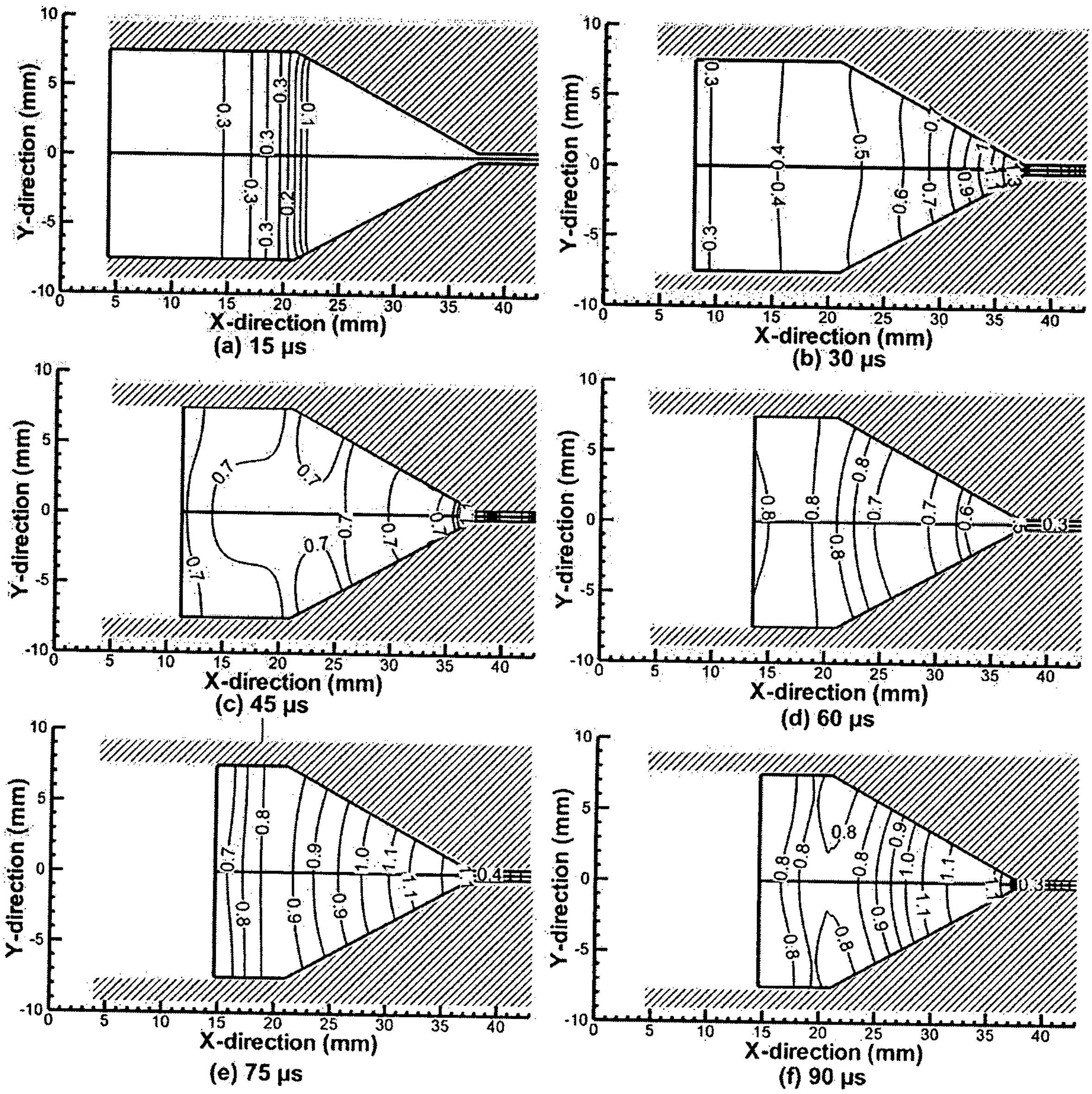


Fig. 12 Pressure profile (GPa) of diesel inside conical nozzle

Simulation results of parabolic nozzle flow behavior generated by momentum exchange method and direct impact method are shown in Fig.13 and Fig.14 respectively. In the simulation, projectile velocity around 414 m/s and diesel liquid are used. We found that the pressure created with momentum exchange method is lower than with other. In addition, the pressure is slightly built up with brass piston movement toward the nozzle inside and wave propagation can not be observed. This is reasonably confirmed to the results shown in Fig 9 as well. Moreover, traveling distance of piston inside the nozzle is very short (around 1.8 mm). It is because the momentum of the projectile is still stored inside the piston body instead of complete transfer. Hence the pressure inside the nozzle cavity is low. However, total momentum of piston attached to projectile equal in other method; thus, it requires more and more duration time of the jetting process to transfer energy at low speed into retained liquid. Reasonably, increasing mass or length of brass piston causes reduction of injection pressure and jet velocity, as shown in Fig 10. This is different in nozzle flow behavior generated with direct impact method. With such method, pressure distribution during the process can be observed. In addition, wave propagations were also captured by CFD simulation as shown in Fig. 14. Some of the momentum of the projectile can be directly released into the liquid at a much shorter time; therefore, extremely high pressure and wave propagation are produced. Injection pressure is increased by wave reflection inside nozzle cavity. This is similar in step nozzle as described in previous section. However those values of injection pressure and pressure profile are different and depended on nozzle shaped and geometry as shown in Fig 11, 12 and 14.

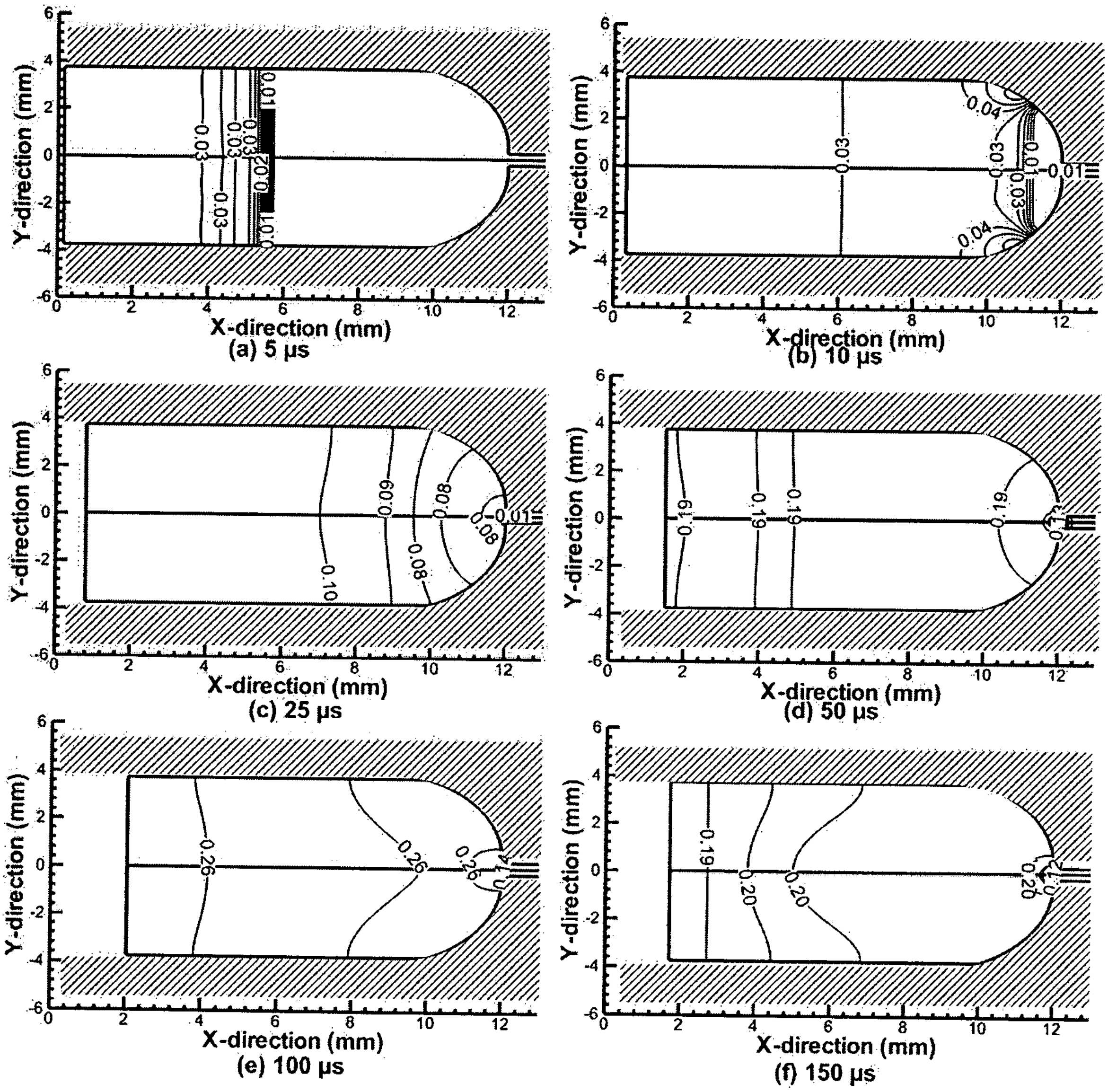


Fig.13 Pressure profile (GPa) created with momentum exchange method

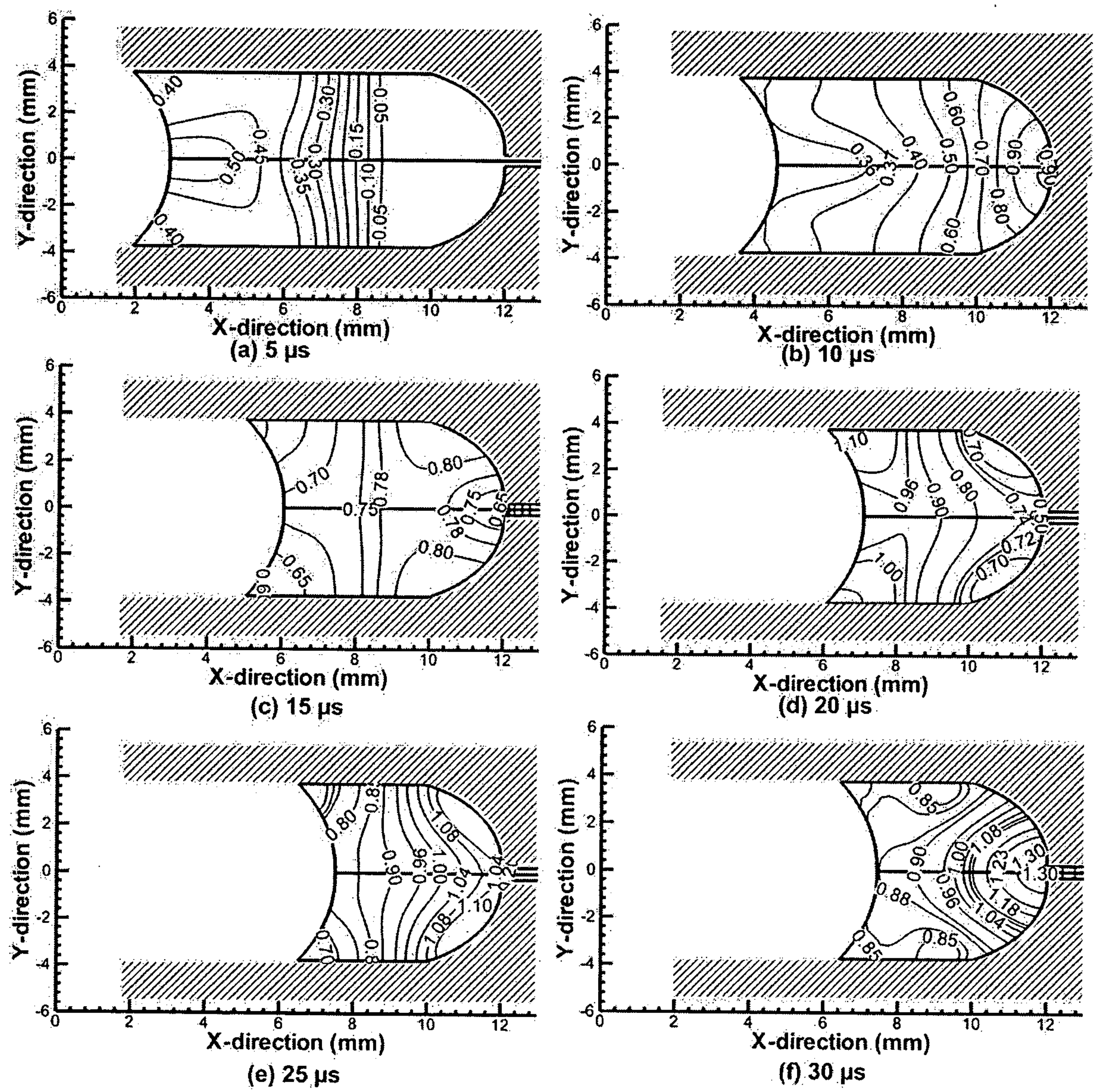


Fig.14 Pressure profile (GPa) created with direct impact method

6. Concluding remarks

In this study, high speed jets is numerically generated by using the impact driven method, from the impact of a high velocity projectile on the liquid package contained in the nozzle cavity. The Computational Fluid Dynamics (CFD) technique is employed for simulation of jet generation process by IDM method within closed system for the firs time. The two fluids model consisting of liquid and air can be successfully calculated. The CFD results show good agreement to the previous experimental results. We found that, for conical nozzle the first reflection peak produces the highest pressure, while in the case of step nozzle the highest pressure peak occurs at the second reflection. Moreover, the momentum exchange rate between projectile and compressed liquid during jet generation process depends significantly on the nozzle cavity geometry more the volume of retained liquid.

Acknowledgments

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.

References

[1] Giudice,E.L. and Campbell,J.D., 2006, "Needle-free vaccine delivery," Advanced Drug Delivery Reviews, Vol. 58, pp. 68 - 89.

[2] Baxter,J.S. and Mitragotri,S., 2009, "Needle-free jet injections: dependence of jet penetration and dispersion in the skin on jet power," Journal of Controlled Release, Vol. 97, pp. 527 - 535.

[3] Bowden,F.P. and Brunton,J.H., 1958, "Damage to solids by liquid impact at supersonic speed," Nature, Vol. 181, pp. 873 - 875.

[4] Keefe,J.D.O., Wrinkle,W.W., and Scully,C.N., 1967, "Supersonic liquid jet," Nature, Vol. 213, pp. 23 - 25.

[5] Ryhm,I.R., 1973, "Analysis of unsteady incompressible jet nozle flow," Journal of Applied Mathematics and Physics, Vol. 24, pp. 149 - 164.

[6] Lesser,M., 1995, "Thirty years of liquid impact research: a tutorial review," Wear, Vol. 186-187, pp. 28 - 34.

[7] Shi,H.H. and Takayama,K., 1995, "Generation of high speed liquid jets by high speed impact of a projectile," JSME International Journal, Vol. 38, pp. 181 - 190.

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

[9] 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.

[10] Pianthong,K., Zakrzewski,S., Behnia,M., and Milton,B.E., 2003, "Characteristics of impact driven supersonic liquid jets," Experimental Thermal and Fluid Science, Vol. 27, pp. 589 - 598.

[11] Zakrzewski,S., Milton,B.E., Pianthong,K., and Behnia,M., 2004, "Supersonic liquid fuel jets injected into quiescent air," International Journal of Heat and Fluid Flow, Vol. 25, pp. 833 - 840.

[12] Pianthong,K., Matthujak,A., Takayama,K., Saito,T., and Milton,B., 2006, "Visualization of supersonic liquid fuel jets," Journal of Flow Visualization and Image Processing, Vol. 13, pp. 217 - 242.

[13] Pianthong,K., Takayama,K., Milton,B., and Behnia,M., 2005, "Multiple pulsed hypersonic liquid diesel fuel jets driven by projectile impact," Shock Waves, Vol. 14, pp. 73 - 82.

[14] Pianthong,K., Matthujak,A., Takayama,K., Miton,B.E., and Behnia,M., 2008, "Dynamic characteristics of pulsed supersonic fuel sprays," Shock Waves, Vol. 18, pp. 1 - 10.

[15] Shi, H.H., 1994, "Study of Hypersonic Liquid Jets" Ph.D. Thesis, Tohoku University.

[16] 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.

[17] Fluent Inc., 2005, "FLUENT 6.2 User's Guide,"

[18] Fluent Inc., 2005, "FLUENT 6.2 UDF Manual,"

VISUALIZATION OF INJECTION PROCESS IN A NEEDLE-FREE INJECTION DEVICE BY HIGH-SPEED VIDEO CAMERA AND CFD

K. Pianthong [1], W. Seehanam [1], W. Sittiwong [1] and B.E. Milton [2]

[1] Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Thailand
[2] School of Mechanical and Manufacturing Engineering, University of New South Wales, Sydney, Australia

ABSTRACT
Jet injection characteristics which are jet velocity, piston movement, and jet penetration in the tissue simulant, for the needle-free jet injection, are thoroughly investigated by using the high-speed video camera and CFD simulation. In the experiments, a high-speed video camera was used to capture the jet flow phenomena in the medium which are the quiescent air and the 20% polyacrylamide gel used as tissue simulant. For the injection into the air, by using experimental and numerical visualization, it is found that liquid volume ejected is decreased, the jet velocity slightly increase, and the average velocity of piston movement during jet injection process is found to be steadily decay over remaining 0.15-0.20 m/s after the high velocity pulse during the first 1-10 ms. The CFD results show good agreement to the results from the experiment both quantitatively and qualitatively. For the injection into 20% polyacrylamide, the camera can capture the penetrative process and can be divided into three stages which are the threshold stage, the penetrative stage, and the dispersion stage.

INTRODUCTION
Jet injector delivers liquid medication or vaccine through a nozzle orifice via a high pressure, high speed narrow stream that penetrates the skin, as shown in Fig. 1. Drug or vaccine can be delivered to intradermal, subcutaneous, or intramuscular tissue depending on operating parameters performed by the jet injector device. The devices designed to deliver the drug were first developed in the 1940's and were widely used for mass immunization campaigns from the 1950's to the 1980's [1-2]. It is believed that jet injection devices should improve the efficacy of the drug, due to better distribution in tissues where liquid drug delivered via jet injection is dispersed more widely; in addition, site pain after injection is very small, in a range of hundred micrometers. This mechanism was confirmed by J. Baxter's studies [3-4]. An important advantage of jet injectors over other novel needle-free drug delivery methods is that parenteral delivery of drug to the same sites as those used in needle and syringe delivery may allow for use of the same vaccine formulations with the same proven efficacy [2].

The device, there are many disadvantage of needle free jet injection device distributed in the market; therefore, the development have been required to improve efficiency of the jet injection method and the device. One study compared two alternative jet injector devices with standard device showed that the jet injector devices were associated with higher levels of pain and more local reactions; moreover, there is blood contamination in head of jet injection device after injection [1, 5]. This is because the device generates the liquid jet at high velocity and impact pressure resulting in blood splashed back from the patient [1, 2, 5]. For this reason, understanding on effects of the parameters on characteristics and behavior of needle-free jet injection for the completely controllable device has been essence to correctly specify the hole depth, created by the jet liquid jet, in the target tissue [2].

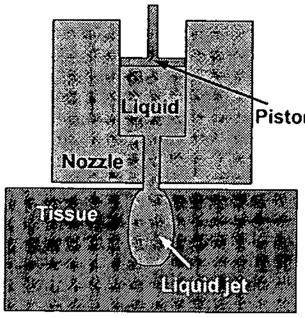


Fig. 1. Needle-free jet injection method

The devices have been concerned with injection efficiency corresponding to operating parameters which are jet penetration depth, liquid dispersion, jet velocity, volume ejected, and nozzle diameter. In works of Joy Schramm-Baxtera *et al.* [3, 6] mentioned that with increasing the nozzle diameter and jet velocity, the shape of liquid dispersion at the end of the hole in simulant tissue is changed and jet penetration depth is increased. Further works [4] from this research group showed that depth of the injection hole increases with ejected volume before reaching an asymptotic volume. In addition, Shergold *et al.* [7] explored the penetration of a soft solid by a liquid jet injection from commercial needle-free jet injection devices, and revealed the discharge characteristic. A high pressure pulse, around 20-35 MPa, during the first 1-5 ms of injection, followed by steady decay in liquid pressure was found. However, those previous studies did not explore the jet generation behavior inside the nozzle during injection process, even though it directly affects on the characteristics of jet injection.

Therefore, in this study, the jet injection parameters, which are jet velocity, piston movement, and jet penetration in 20% polyacrylamide, for the needle-free jet injection, are thoroughly investigated by using the high-speed video camera and CFD simulation for the better understanding of needle-free jet injection process and providing useful information in the design of needle -free jet injection apparatus.

MATERIAL AND METHOD

Experimental setup

Liquid jets, in these experiments, were produced from a commercial, spring-driven growth hormone jet injector, Cool Click (Bioject2000 Inc.) through an orifice of 0.17 mm in diameter. The maximum liquid volume ejected was 0.5 ml. In experiment, the jet is injected into the air and 20 % polyacrylamide gel, in which a nozzle tip attached the gel during the injection, and deionized (DI) water was used as the jet fluid. Optical setup, as shown in Fig. 2 was applied to visualize the jet flow phenomena and penetrative process in the gel. To gain a better understanding of the dynamics characteristics in time series, a high-speed video camera (Photron Fastcam SA5) was used to capture the jet flow phenomena in the medium and piston behavior during a jet ejection. During the operation process of the needle-free injection device, the frame rate of 15,000 frames per second (fps) was applied, and the major operating parameters, which are jet velocity, jet penetration, and piston movement is thoroughly investigated.

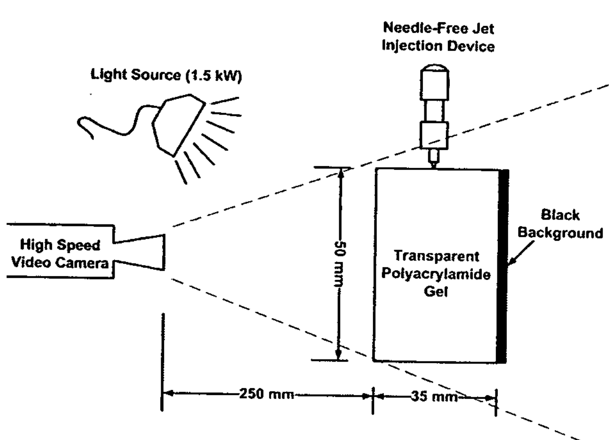


Fig. 2. Visualization setup

'olyacrylamide gels

0% Polyacrylamide gel were used as a model soft ıaterial which Young's modulus and hardness, reported y Schramm-Baxter et al. [6] are 0.22 MPa and 41 Hoo, ɜspectively. The 20% gel was created by the addition of ıitiators (10% ammonium persulphate (APS) and [,N,N',N'-tetramethylethylenediamine (TEMED)) to a 0% (w/v) acrylamide solution. The acrylamide solution 'as mixed with DI water to create solutions possessing ɔrylamide concentrations in the range of 20% w/v, and ıe gel was polymerized by the addition of 60 ul 10% APS ıd 12 ul TEMED to the acrylamide solution of 6 ml.

'FD modeling

rom the mechanism of needle-free jet injection process, ıis setup can be modeled in a closed system domain with ĸi-symmetric geometry divided into two zone: nozzle ıvity zone being full of dose liquid and air zone, as ıown in Fig. 3. The CFD commercial code (FLUENT) is ıed as the tool to simulate the dynamic characteristics of e jet generation process. In the simulation, the two-fluid model consisting of liquid and air can be calculated by using the volume of fraction (VOF) model for interaction between fluid jet and air. The air and liquid density are simply specified to be compressible fluid by using the formula of ideal gas and compressible liquid including the instant liquid density, respectively. The turbulence model is the standard k-ε model with segregate solver for the non-linear equations. The velocity of the piston movement assuming as a moving wall during the injection is computed from the resulting force from the combination of spring, pressure, and friction forces acting on the piston in x-direction. The initial spring force can be calculated from Hooke's law equation where the spring force constant tested by using Rimac Spring Tester is around 17.8 kN/m.

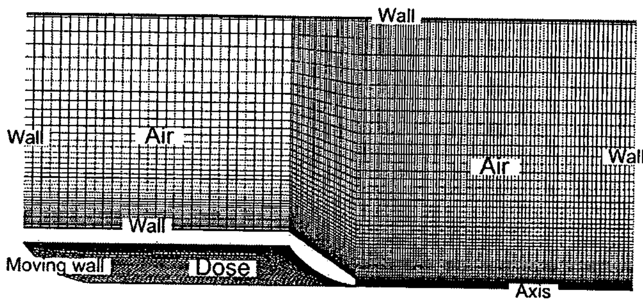


Fig. 3. CFD Modeling domain

RESULTS AMD DISCUSSION

Characteristics of the jet generation process

Dynamics characteristics of the piston and the jet injected into quiescent air, expressed as average velocities defined as its penetration distance along the medium divided by emerging time, are shown in Fig. 4 and 5. These figures show the results from the experiment and simulation. It is observed that water volume ejected is decreased, the jet velocity slightly increase, as shown in Fig. 4, and average velocity of piston movement during jet injection process, as shown in Fig. 5, is found to be steadily decay over remaining 0.15 – 0.20 m/s, before there is a high velocity pulse during the first 1-10 ms. This corresponds to the discharge characteristics, expressed as stagnation pressure, which was found by Shergold *et al.* [7]. However, it is observed that the average jet velocity trends from the both method are only slightly different; although, the CFD simulation gives higher average velocities than those from experiments. In the simulation, the phenomenon of the atomization is specified by simply VOF two phase flow model, and this causes the spray atomization, corresponding to dynamics drag, occurring jet injection into the air is not fully taken into account in the CFD model. This is in accord with the results shown in Fig 6. The thin jet can be found in CFD results while cloud of liquid atomization occurring around the jet during jet injection can be captured by a high-speed video camera.

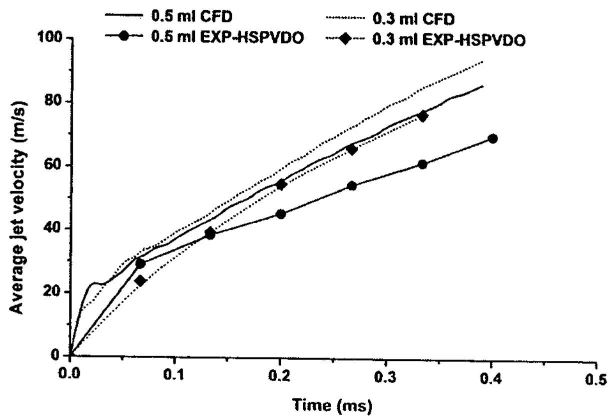


Fig. 4. Average jet velocities in quiescent air

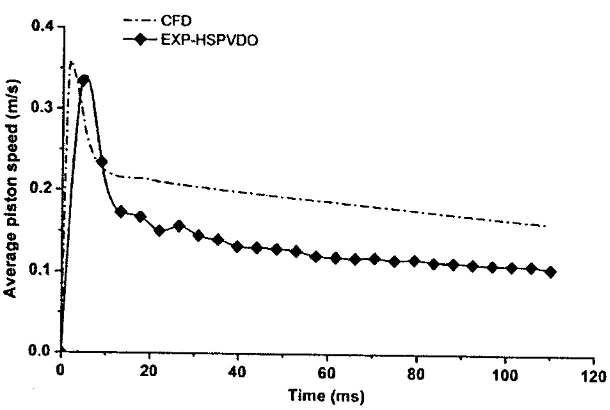


Fig. 5. Average piston speed

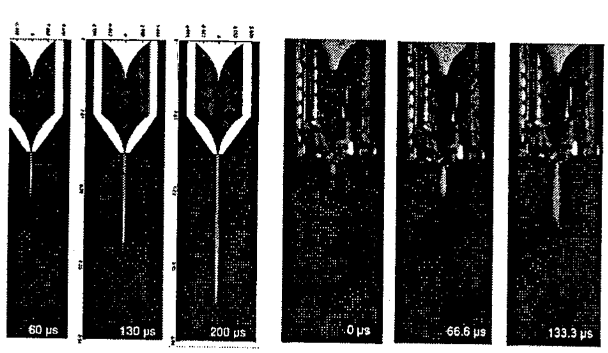


Fig. 6. Visualization of the jet ejected into the air

:haracteristics of the penetrative process

'rom the experiment results, characteristics of penetrative ırocess of the jet during the injection into polyacrylamide ;el used as tissue stimulant are shown in Fig. 7 to Fig. 10. 'ig. 7 shows the high speed video camera images for the ıenetration of the jet in 20% polyacrylamide, which the '.4 ml volume of liquid retained inside a nozzle was used. 'rom the visualization, three stages of the penetrative rocess can be observed. The first are the threshold stage /hich starts at initial movement of a compressing piston. 'he jet did not pierce through the gel surface (as shown in ig. 7a-7j) because the impact pressure and the ıomentum of the jet at the initial stage did not exceed the ırface threshold pressure of the gel. It is note that the short time of the threshold stage can be achieved with the low volume of ejected liquid as shown in Fig. 8 which presents penetration of 0.2 ml liquid jet. For the second stage (penetrative stage), after the appropriate impact pressure and momentum, the jet can pierces through the gel surface and quickly penetrates the polyacrylamide gel as shown in Fig 7k-7l. In this stage, few dispersion of liquid and narrow hole can be observed. The last stage of the process is the dispersion stage which the penetrative velocity slightly is decreased while boundary of the liquid injected initially expands as shown in Fig 7m – 7u. This stage continuously operates along the injection process until completion.

Usually, the penetrative process of jet injection mainly depends on the volume of retained liquid in a nozzle. Therefore, this study presents the influence of such volume on the penetration behavior. With volume of ejected water raging from 0.1 to 0.5 ml, Fig. 9 and 10 shows the averaged velocity and distance of liquid penetration. The velocity and distance correspond to the penetrative phenomena in Fig 7 and 8; the three stages of the penetrative process can be also found. At the threshold stage the average velocities is very high and suddenly decrease at initial injection before those velocities is quite constant as shown in Fig 9. This significantly relates to penetration distance which is reasonably constant at first 0 - 1 ms. It is note that the long time of the threshold stage can be achieved with the high volume of ejected liquid. This is because the momentum and velocity of the jet ejected from high liquid volume are small as described in the previous subsection. After that penetrative velocity is increased because the liquid can pierce through the gel surface and penetration distance more extends. This is at penetrating stage. Highest penetrative velocities in the injection process decrease with high ejected liquid volume. Because of the largely penetrated hole and high dispersion rate due to high injected mass. For the last as dispersion stage, the velocities slightly decrease to disperse the liquid into the medium.

CONCLUDING REMARKS

In this study, the jet injection parameters, which are jet velocity, piston movement, and jet penetration in the tissue simulant, for the needle-free jet injection, are thoroughly investigated by using the high-speed video camera and CFD simulation. For the injection into the air, by using experimental and numerical visualization, it is found that liquid volume ejected is decreased, the jet velocity slightly increase, and the average velocity of piston movement during jet injection process is found to be steady decay over remaining 0.15 – 0. 2 m/s after the high velocity pulse during the first 1-10 ms. The CFD results show good agreement to the results from the experiment both quantitatively and qualitatively. For the injection into 20% polyacrylamide, the camera can capture the penetrative process which can be divided into three stages. These are the threshold stage, the penetrative stage, and the dispersion stage.

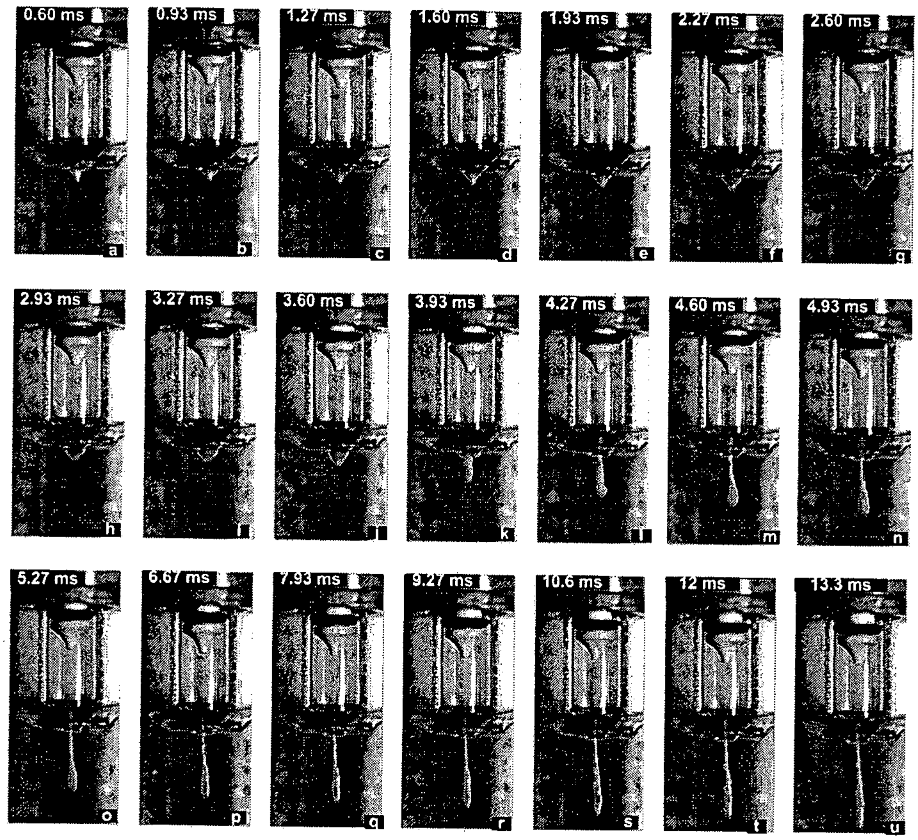


Fig. 7. Visualization of the 0.4 ml liquid jet ejected into polyacrylamide

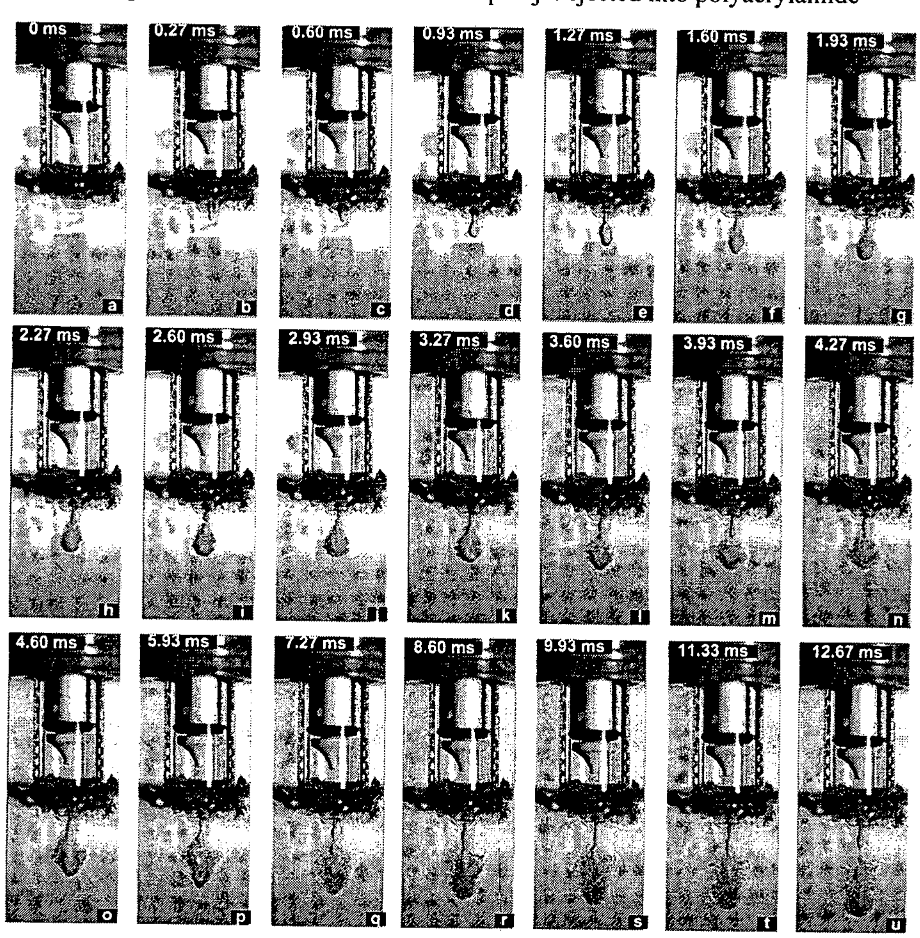


Fig. 8. Visualization of the 0.2 ml liquid jet ejected into polyacrylamide

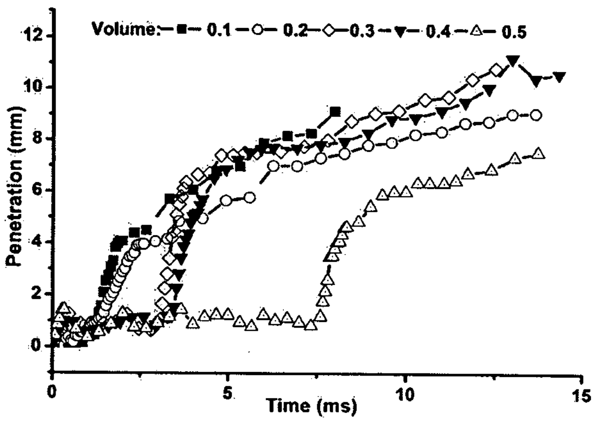


Fig. 9. Velocity of jet penetrating in polyacylamide

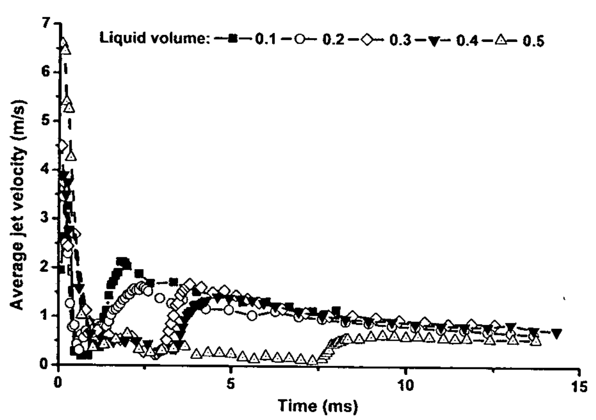


Fig. 10. Penetration of liquid jet in polyacylamide

ACKNOWLEDGMENT

This work was granted by the Office of the Higher Education Commission and Thailand Research Fund (RTF), contract N0. RMU5180020, the National Research Council of Thailand (NRCT) through Ubon Ratchatani University Research Grant fiscal year 2007. Wirapan Seehanam was supported by CHE Ph.D. Scholarship.

REFERENCES

[1] Giudice, E. L. and Campbell, J. D.: *Adv. Drug Delivery Rev.*, Vol. 58 (2006), 68.

[2] O'Hagan, D.T. and Rappuoli, R.: *Adv. Drug Delivery Rev.*, Vol. 58 (2006), 29.

[3] Baxter, J. and Mitragotri, S.: *J. Controlled Release*, Vol. 97 (2004), 517.

[4] Baxter, J. and Mitragotri, S.: *J. Controlled Release*, Vol. 106 (2005), 361.

[5] Jackson, L. A., Robert, G. A., Chen, T., Stout, R. and DeStefano, F., Gorse, G. J., Newman, F. K., Yu, O. and Weniger, B. G.: *Vaccine*, Vo.19, (2001), 4703.

[6] Baxter, J., Katrencik, J., Mitragotri, S.: *J. Biomech.*, Vol. 37 (2004), 1181.

[7] Shergold, O. A., Fleck, N. A. and King, T. S.: *J. Biomech.*, Vol. 39 (2006), 2593.

Injection Characteristics of Liquid Jet from a Needle Free Injection Device in the Tissue Simulant

Wirapan Seehanam[1,*], Kulachate Pianthong[1], Wuttichai Sittiwong[1] and Brian Milton[2]

[1] Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Thailand, 34190
[2] School of Mechanical and Manufacturing Engineering, University of New South Wales, Sydney, Australia, 2052
*Corresponding Author: E-mail: K.Pianthong@ubu.ac.th, Tel: +66-4535-3309, Fax: +66-4535-3308

Abstract

This study aims to investigate the dynamics characteristics of liquid jet injected from a needle-free injection device by analyzing the the flow visualization from the high-speed video camera and the CFD calculation. This is to investigate the jet flow phenomena (e.g. penetration, dispersion, velocity) in the quiescent air and in the tissue simulant (20% polyacrylamide gel) as the further information to apply in the real tissues. The jet injection parameters, which are jet velocity, piston movement, and jet penetration in the tissue simulant, for the needle-free jet injection, are thoroughly investigated by using the high-speed video camera and CFD simulation. In the experimental visualization, a high-speed video camera was used to capture the jet flow phenomena in the medium which are the quiescent air and the 20% polyacrylamide gel used as tissue simulant. Jet injection into the air, both in an experimental and numerical visualization, it is found that when the liquid volume ejected is decreased, the jet velocity slightly increases, and the average velocity of piston movement during jet injection process is found to be steadily decay over remaining 0.15 – 0. 2 m/s after the high velocity pulse during the first 1-10 ms. The CFD results show good agreement to results from experiments both quantitatively and qualitatively. Injecting into 20% polyacrylamide, the jet can be captured by the high speed video camera. The penetration process which consists of three stages which are the threshold stage, the penetrative stage, and the dispersion stage can be revealed.

Keywords: needle – free jet injection, CFD, tissue simulant

1. Introduction

Jet injector delivers liquid medication or vaccine through a nozzle orifice via a high pressure, high speed narrow stream that penetrates the skin, as shown in Fig. 1. Drug or vaccine can be delivered to intradermal, subcutaneous, or intramuscular tissue depending on operating parameters performed by the jet injector device. The devices designed to deliver the drug were first developed in the 1940's and were widely used for mass immunization campaigns from the 1950's to the 1980's [1-2]. It is believed that jet injection devices should improve the efficacy of the drug, due to better distribution in tissues where liquid drug delivered via jet injection is dispersed more widely; in addition, site pain after injection is very small, in a range of hundred micrometers. This mechanism was confirmed by J. Baxter's studies [3-4]. An important advantage of jet injectors over other novel needle-free drug delivery methods is that parenteral delivery of drug to the same sites as those used in needle and syringe delivery may allow for use of the same vaccine formulations with the same proven efficacy [2].

The device, there are many disadvantage of needle free jet injection device distributed in the market; therefore, the development have been required to improve efficiency of the jet injection method and the device. One study compared two alternative jet injector devices with standard device showed that the jet injector devices were associated with higher levels of pain and more local reactions; moreover, there is blood contamination in head of jet injection device after injection [1, 5]. This is because the device generates the liquid jet at high velocity and impact pressure resulting in blood splashed back from the patient [1, 2, 5]. For this reason, understanding on effects of the parameters on characteristics and behavior of needle-free jet injection for the completely controllable device has been essence to correctly specify the hole depth, created by the jet liquid jet, in the target tissue [2].

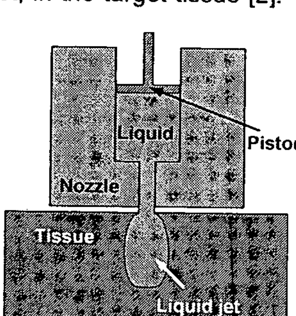


Fig. 1 Needle-free jet injection method

The devices have been concerned with injection efficiency corresponding to operating parameters which are jet penetration depth, liquid dispersion, jet velocity, volume ejected, and nozzle diameter. Jet power (Po) has been introduced as a combined parameter for those describing dependences and is calculated as:

$$P_o = \frac{1}{8}\pi\rho D_0^2 u_0^3 \qquad (1)$$

where D_0 is a nozzle diameter, u_0 is an exit velocity and ρ is a liquid density. In works of Joy Schramm-Baxtera et al. [3, 6] mentioned that with increasing the jet power, the shape of liquid dispersion at the end of the hole in simulant tissue is changed and jet penetration depth is increased. Further works [4] from this research group showed that depth of the injection hole increases with ejected volume before reaching an asymptotic volume. In addition, Shergold et al. [7] explored the penetration of a soft solid by a liquid jet injection from commercial needle-free jet injection devices, and revealed the discharge characteristic. A high pressure pulse, around 20-35 MPa, during the first 1-5 ms of injection, followed by steady decay in liquid pressure was found. However, those previous studies did not explore the jet generation behavior inside the nozzle during injection process, even though it directly affects on the characteristics of jet injection.

Therefore, in this study, the jet injection parameters, which are jet velocity, piston movement, and jet penetration in 20% polyacrylamide, for the needle-free jet injection, are thoroughly investigated by using the high-speed video camera and CFD simulation for the better understanding of needle-free jet injection process and providing useful information in the design of needle -free jet injection apparatus.

2. Material and Method

2.1 Experimental setup

Liquid jets, in these experiments, were produced from a commercial, spring-driven growth hormone jet injector, Cool Click (Bioject2000 Inc.) through an orifice of 0.17 mm in diameter. The maximum liquid volume ejected was 0.5 ml. In experiment, the jet is injected into the air and 20 % polyacrylamide gel, in which a nozzle tip attached the gel during the injection, and deionized (DI) water was used as the jet fluid. Optical setup, as shown in Fig. 2 was applied to visualize the jet flow phenomena and penetrative process in the gel. To gain a better understanding of the dynamics characteristics in time series, a high-speed video camera (Photron Fastcam SA5) was used to capture the jet flow phenomena in the medium and piston behavior during a jet ejection. During the operation process of the needle-free injection device, the frame rate of 15,000 frames per second (fps) was applied, and the major operating parameters, which are jet velocity, jet penetration, and piston movement is thoroughly investigated.

2.2 Tissue simulant

20% Polyacrylamide gel were used as a model soft material which Young's modulus and hardness, reported by Schramm-Baxter et al. [6] are 0.22 MPa and 41 Hoo, respectively. The 20% gel was created by the addition of initiators (10% ammonium persulphate (APS) and N,N,N',N'-tetramethylethylenediamine (TEMED)) to a 40% (w/v) acrylamide solution. The acrylamide solution was mixed with DI water to create solutions possessing acrylamide concentrations in the range of 20% w/v, and the gel was polymerized by the addition of 60 ul 10% APS and 12 ul TEMED to the acrylamide solution of 6 ml.

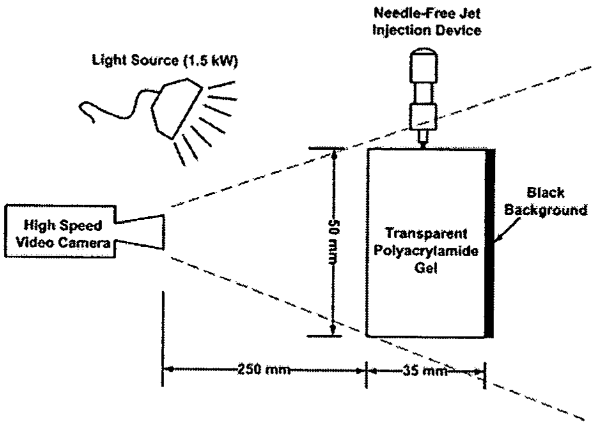


Fig. 2 Visualization setup

2.3 CFD modeling

From the mechanism of needle-free jet injection process, this setup can be modeled in a closed system domain with axi-symmetric geometry divided into two zone: nozzle cavity zone being full of dose liquid and air zone, as shown in Fig. 3. The CFD commercial code (FLUENT) is used as the tool to simulate the dynamic characteristics of the jet generation process. In the simulation, the two-fluid model consisting of liquid and air can be calculated by using the volume of fraction (VOF) model for interaction between fluid jet and air. The air and liquid density are simply specified to be compressible fluid by using the formula of ideal gas and compressible liquid including the instant liquid density, respectively. The turbulence model is the standard k-ε model with segregate solver for the non-linear equations. The velocity of the piston movement assuming as a moving wall during the injection is computed from the resulting force from the combination of spring, pressure, and friction forces acting on the piston in x-direction. The initial spring force can be calculated from Hooke's law equation where the spring force constant tested by using Rimac Spring Tester is around 17.8 kN/m.

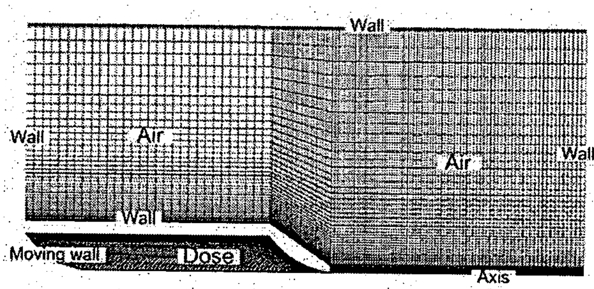


Fig. 3 CFD Modeling domain

4. Result and Discussion

4.1 Characteristics of the jet generation process

Dynamics characteristics of the piston and the jet injected into quiescent air, expressed as average velocities defined as its penetration distance along the medium divided by emerging time, are shown in Fig. 4 and 5. These figures show the results from the experiment and simulation. It is observed that water volume ejected is decreased, the jet velocity slightly increase, as shown in Fig. 4, and average velocity of piston movement during jet injection process, as shown in Fig. 5, is found to be steadily decay over remaining 0.15 – 0.20 m/s, before there is a high velocity pulse during the first 1-10 ms. This corresponds to the discharge characteristics, expressed as stagnation pressure, which was found by Shergold et al. [7]. However, it is observed that the average jet velocity trends from the both method are only slightly different; although, the CFD simulation gives higher average velocities than those from experiments. In the simulation, the phenomenon of the atomization is specified by simply VOF two phase flow model, and this causes the spray atomization, corresponding to dynamics drag, occurring jet injection into the air is not fully taken into account in the CFD model. This is in accord with the results shown in Fig 6. The thin jet can be found in CFD results while cloud of liquid atomization occurring around the jet during jet injection can be captured by a high-speed video camera.

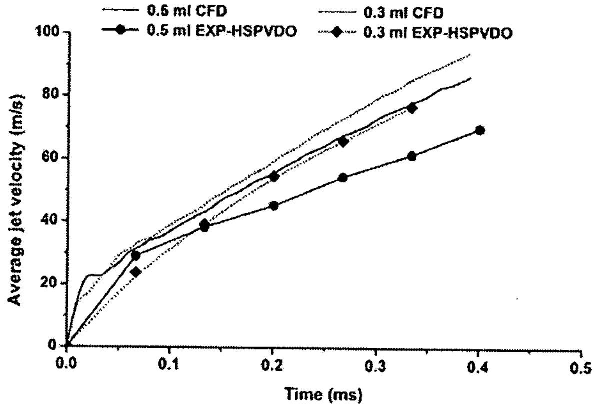


Fig. 4 Average jet velocities in quiescent air

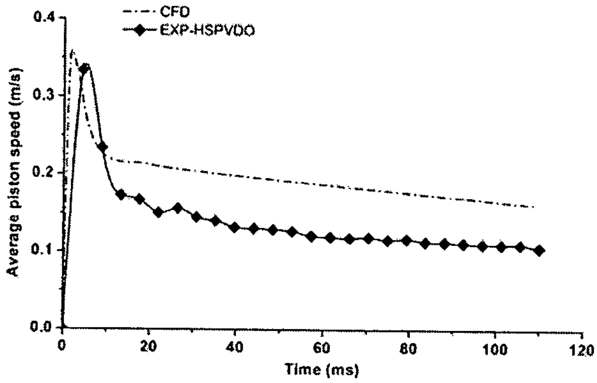


Fig. 5 Average piston speed

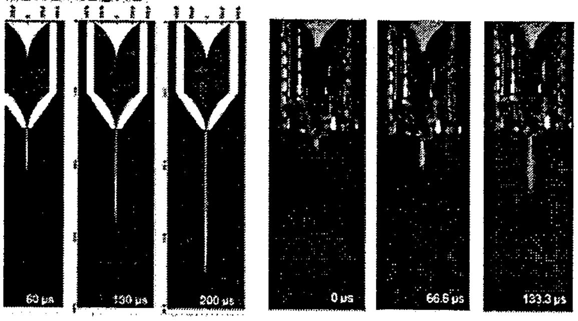


Fig. 6 Visualization of the jet ejected into the air

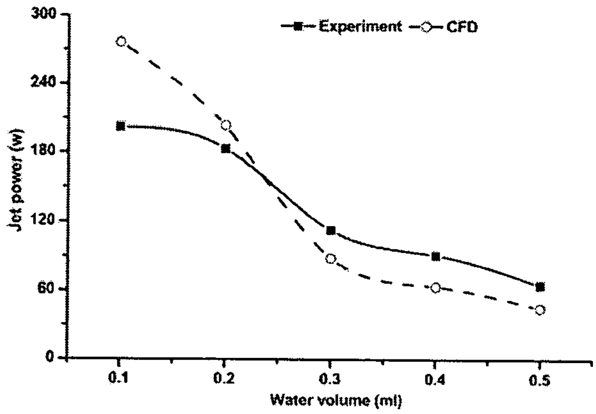


Fig 7 Jet power at various water volumes

Fig 7 shows the effect of water volume on the jet power calculated by using equation (1) and the exit velocity resulted from experiment and CFD simulation methods. In both methods, it is observed that water volume ejected is increased, the jet power slightly decrease. This is because the exit jet velocity is decreased with the high injected volume. The initial injection pressure built up by first shock reflection near the nozzle exit is low with the high volume of column liquid. In addition, with the volume between 0.1 – 0.2 ml, the jet power given by CFD calculation is higher than by experiments. In contrasts to those injection volumes, experiment gives the results with higher power than the CFD result. The differences between the CFD and experiment results may come from the disintegration or atomization of the jet which is not fully taken into account in the CFD yet.

4.2 Characteristics of the penetrative process

From the experiment results, characteristics of penetrative process of the jet during the injection into polyacrylamide gel used as tissue stimulant are shown in Fig. 8 to Fig. 11. Fig. 8 shows the high speed video camera images for the penetration of the jet in 20% polyacrylamide, which the 0.4 ml volume of liquid retained inside a nozzle was used. From the visualization, three stages of the penetrative process can be observed. The first are the threshold stage which starts at initial movement of a compressing piston. The jet did not pierce through the gel surface (as shown in Fig. 8a-8j) because the impact pressure and the momentum of the jet at the initial stage did not exceed the surface threshold pressure of the gel. It is note that the short time of the threshold stage can be achieved with the low volume of ejected liquid as shown in Fig. 9 which presents penetration of 0.2 ml liquid jet. For the second stage (penetrative stage), after the appropriate impact pressure and momentum, the jet can pierces through the gel surface and quickly penetrates the polyacrylamide gel as shown in Fig 8k-8l. In this stage, few dispersion of liquid and narrow hole can be observed. The last stage of the process is the dispersion stage which the penetrative velocity slightly is decreased while boundary of the liquid injected initially expands as shown in Fig 8m – 8u. This stage continuously operates along the injection process until completion.

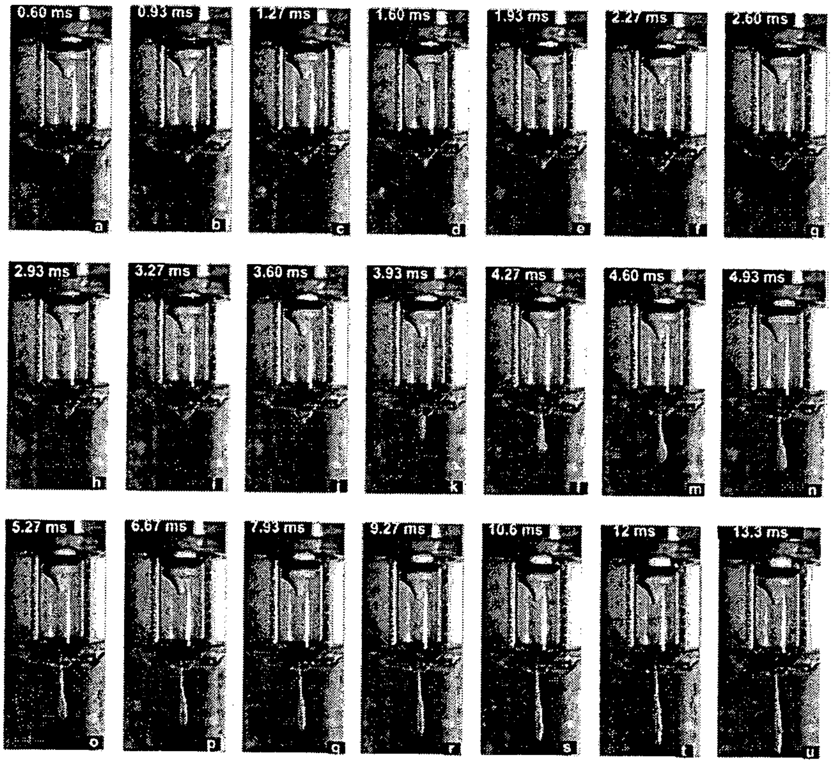


Fig. 8 Visualization of the 0.4 ml liquid jet (43.14 W of jet power) ejected into polyacrylamide

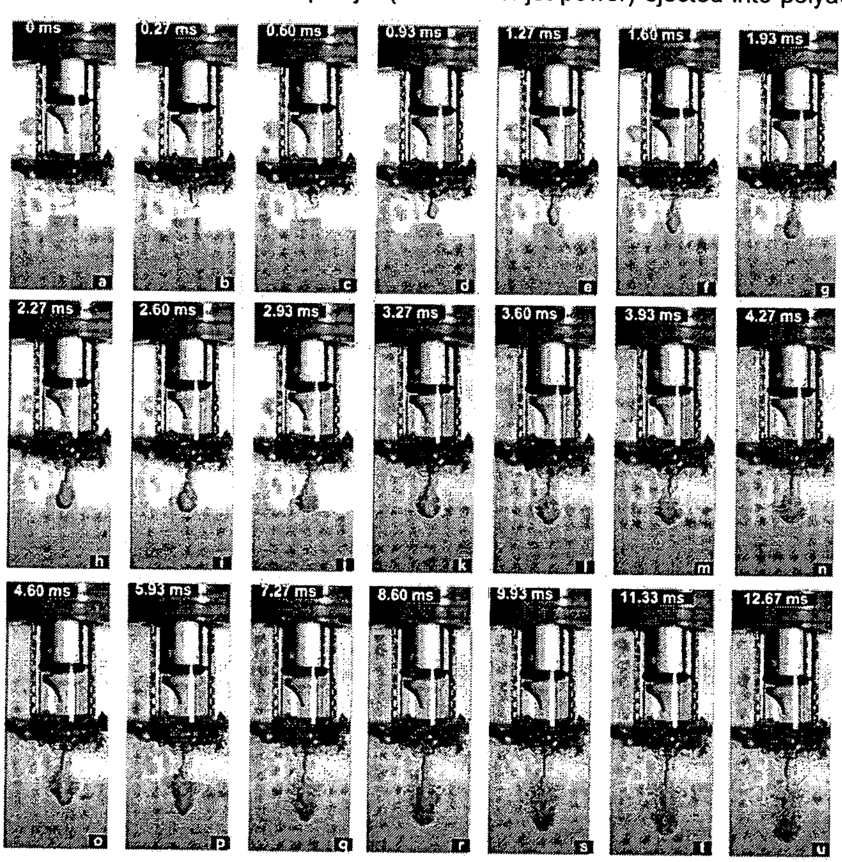


Fig. 9 Visualization of the 0.2 ml liquid jet (55.44 W of jet power) ejected into polyacrylamide

Usually, the penetrative process of jet injection mainly depends on the volume of retained liquid in a nozzle. Therefore, this study presents the influence of such volume on the penetration behavior. With volume of ejected water raging from 0.1 to 0.5 ml, Fig. 10 and 11 shows the averaged velocity and distance of liquid penetration. The velocity and distance correspond to the penetrative phenomena in Fig 8 and 9; the three stages of the penetrative process can be also found. At the threshold stage the average velocities is very high and suddenly decrease at initial injection before those velocities is quite constant as shown in Fig 10. This significantly relates to penetration distance which is reasonably constant at first 0 - 1 ms. It is note that the long time of the threshold stage can be achieved with the high volume of ejected liquid. This is because the momentum and velocity of the jet ejected from high liquid volume are small as described in the previous subsection. After that penetrative velocity is increased because the liquid can pierce through the gel surface and penetration distance more extends. This is at penetrating stage. Highest penetrative velocities in the injection process decrease with high ejected liquid volume. Because of the largely penetrated hole and high dispersion rate due to high injected mass. For the last as dispersion stage, the velocities slightly decrease to disperse the liquid into the medium.

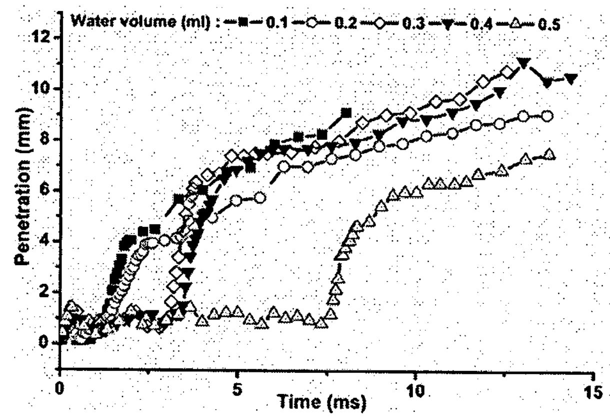


Fig. 10 Velocity of jet penetrating in polyacylamide

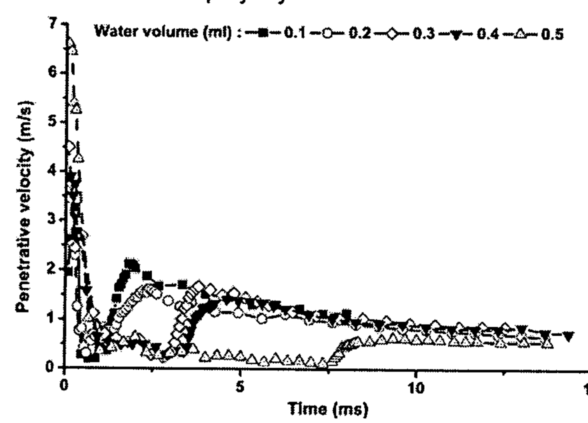


Fig. 11 Penetration of liquid jet in polyacylamide

5. Concluding remarks

In this study, the jet injection parameters, which are jet velocity, piston movement, and jet penetration in the tissue simulant, for the needle-free jet injection, are thoroughly investigated by using the high-speed video camera and CFD simulation. For the injection into the air, by using experimental and numerical visualization, it is found that liquid volume ejected is decreased, the jet velocity slightly increase, and the average velocity of piston movement during jet injection process is found to be steady decay over remaining 0.15 – 0. 2 m/s after the high velocity pulse during the first 1-10 ms. The CFD results show good agreement to the results from the experiment both quantitatively and qualitatively. For the injection into 20% polyacrylamide, the camera can capture the penetrative process which can be divided into three stages. These are the threshold stage, the penetrative stage, and the dispersion stage.

6. Acknowledgement

This work was granted by the Office of the Higher Education Commission and Thailand Research Fund (RTF), contract NO. RMU5180020, the National Research Council of Thailand (NRCT) through Ubon Ratchatani University Research Grant fiscal year 2007. Wirapan Seehanam was supported by CHE Ph.D. Scholarship.

7. References

[1] Giudice, E. L. and Campbell, J. D. (2006). Needle-free vaccine delivery, *Advanced Drug Delivery Reviews*, vol. 58, March 2006, pp. 68 - 89.

[2] O'Hagan, D.T. and Rappuoli, R. (2006). Novel approaches to pediatric vaccine delivery, *Advanced Drug Delivery Reviews*, vol.58, February 2006, pp. 29-51.

[3] Baxter, J. and Mitragotri, S. (2004). Needle-free jet injections: dependence of jet penetration and dispersion in the skin on jet power, *Journal of Controlled Release*, vol.97, April 2004, pp. 517-535.

[4] Baxter, J. and Mitragotri, S. (2005). Jet-induced skin puncture and its impact on needle-free jet injections: Experimental studies and a predictive model, *Journal of controlled release*, vol. 106, July 2005, pp. 361-373.

[5] Jackson, L. A., Robert, G. A., Chen, T., Stout, R. and DeStefano, F., Gorse, G. J., Newman, F. K., Yu, O. and Weniger, B. G. (2001). Safety and immunogenicity of varying dosages of trivalent inactivated influenza vaccine administered by needle-free jet injectors, *Vaccine*, vol.19, June 2001, pp. 4703 - 4709.

[6] Baxter, J., Katrencik, J., Mitragotri, S. (2004). Jet injection into polyacrylamide gels: investigation of jet injection mechanics, *Journal of Biomechanics*, vol. 37, December 2004, pp. 1181-1188.

[7] 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, August 2006, pp. 2593 - 2602.

Shock Waves - An International Journal on Shock Waves, Detonations and Explosions

Investigation on the generation process of impact driven high speed liquid jets using a CFD technique

--Manuscript Draft--

Manuscript Number:	
Full Title:	Investigation on the generation process of impact driven high speed liquid jets using a CFD technique
Article Type:	Original paper
Keywords:	impact driven liquid jets; liquid jet CFD; shock reflection in liquids
Corresponding Author:	Kulachate Pianthong, Ph.D. Ubon Ratchathani University Warinchamrab, Ubonratchathani THAILAND
Corresponding Author Secondary Information:	
Corresponding Author's Institution:	Ubon Ratchathani University
Corresponding Author's Secondary Institution:	
First Author:	Wirapan Seehanam, Ph.D.Candidate
First Author Secondary Information:	
Order of Authors:	Wirapan Seehanam, Ph.D.Candidate Kulachate Pianthong, Ph.D. Wuttichai Sittiwong, Ph.D.Candidate Brian Edwards Milton, Ph.D. Kazuyoshi Takayama, Ph.D.
Order of Authors Secondary Information:	
Abstract:	High speed liquid jets have applications in many fields of engineering, science and medicine. Therefore, the investigation of their characteristics by modern and inexpensive methods will benefit all these areas. Previously, high speed liquid jets have been studied experimentally using the momentum exchange method, called the "Impact Driven Method (IDM)" by which the impact of a high velocity projectile on the liquid package contained in the nozzle cavity produced the jet. The shock pulse reflection in the cavity caused by the impact then drove a multiple pulsed jet from the nozzle exit. In this study, a Computational Fluid Dynamics (CFD) technique has been employed to simulate the jet generation process by the IDM method within a closed domain. In the simulation, a two fluid simulation consisting of liquid and air can be successfully calculated by using a two phase flow mixture model and a moving mesh for the projectile motion. The CFD results show good agreement to the results of previous experimental studies both quantitatively and qualitatively. In addition, the simulation results capture the wave propagation within the liquid in the nozzle, demonstrating for the first time the dynamic characteristics of multiple pulsed high speed liquid jets initiated by the impact driven method. This provides a breakthrough in the simulation of the supersonic injection of a liquid into air and will be useful fundamental knowledge for future studies of high speed injection and its related fields.
Suggested Reviewers:	Tsutomu Saito, Ph.D. Professor , Muroran Institute of Technology saito@mmm.muroran-it.ac.jp
	Chau Kwong Wing, Ph.D. Professor, The Hong Kong Polytechnic University

	cekwchau@polyu.edu.hk
	Perter Voinovich, Ph.D. Professor, Russian academy of science vpeter@scc.ioffe.ru
	Hong Hui Shi, Ph.D. Professor, Zhejiang Sci-Tech University hhshi@zstu.edu.cn
	Gopalan Jagadeesh, Ph.D. Professor, Indian Institute of Science jaggie@aero.iisc.ernet.in

Shock Waves manuscript No.
(will be inserted by the editor)

[nvestigation on the generation process of impact driven high ;peed liquid jet using CFD technique

W.Seehanam · K.Pianthong · W.Sittiwong · B.E.Milton · K.Takayama

leceived: date / Accepted: date

\bstract High speed liquid jets have applications in nany fields of engineering, science and medicine. There- ɔre, the investigation of their characteristics by modern .nd in-expensive methods will benefit all these areas. 'reviously, high speed liquid jets have been studied ex- ›erimentally using the momentum exchange method, alled the "Impact Driven Method (IDM)", by which he impact of a high velocity projectile on the liquid ›ackage contained in the nozzle cavity produced the ɜt. The shock pulse reflection the cavity caused by the npact then drove a multiple pulsed jet from the noz- le exit. In this study, a Computational Fluid Dynamics CFD) technique has been employed to simulate the jet eneration process by the IDM method within a closed omain. In the simulation, a two-fluid model consist- ıg of liquid and air can be successfully calculated by sing a two phase flow mixture model and a moving ıesh for the projectile motion. The CFD results show ɔod agreement to the results of previous experimental ;udies both quantitatively and qualitatively. In addi- on, the simulation results capture the wave propaga- on within the liquid in the nozzle, proving for the first me the dynamic characteristics of multiple pulsed high ɔeed liquid jets initiated by the impact driven method. 'his provides a breakthrough in the simulation of the supersonic injection of a liquid into air and will be useful fundamental knowledge for future studies of high speed injection and its related fields.

/. Seehanam · K. Pianthong · W. Sittiwong
epartment of Mechanical Engineering,Faculty of Engineer- g, Ubon Ratchathani University, Thailand
-mail: K.Pianthong@gmail.com

.E. Milton
:hool of Mechanical and Manufacturing Engineering, Uni- :rsity of New South Wales, Sydney, Australia

. Takayama
.terdisciplinary Shock Wave Research Center, Institute of luid Science, Tohoku University, Sendai, Japan

Keywords High speed liquid jets · Impact driven method · Computational Fluid Dynamics (CFD) · Shock reflection

1 Introduction

The characteristics of high speed liquid jets have been studied over a number of years, fundamental knowledge of such jets being essential for appropriate application to many industrial technologies such as cutting, drilling, mining, and tunneling. In combustion processes, more- over, fuel sprayed at high speed into air is beneficial in improving combustion, some applications being SCRAM (supersonic combustion RAM) and direct injection diesel engines. This is because the atomization and mixing are enhanced and, if the injection is supersonic, the bow shock wave is likely to provide significantly increased local air temperatures. For medical engineering, with drug injection, the needle may be replaced with a high speed liquid jet to deliver the drug through the skin, this being termed "needle-free jet injection". This de- livery improves the activation because the drug solution can be a very small particle thereby increasing the sur- face of interaction between it and the tissue. It also prevents infection in both the patient and the adminis- trator which can be caused by contaminated injection. In addition, the diameter of the hole after injection with a high speed liquid jet is very small and the scar can heal faster [1,2]. For drug delivery, the liquid jet velocity should be limited to around 100 - 200 m/s, depending on the design condition, which is very much less than hose in combustion technologies. However, in both ap- plications, the high speed liquid jet can be generated by the same method called the "impact driven method or impulsive method".

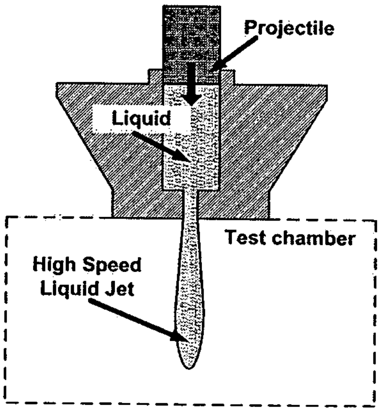
The Impact Driven Method (IDM) technique first presented by Bowden and Brunton in 1958 [3] is a method or producing a high speed liquid jet. The liquid con- ained in nozzle cavity is driven by the impact of a high speed solid projectile which accelerates it to a high velocity. Liquid flow behavior in the liquid nozzle sac during the jet generation process directly affects the haracteristics of the emerging high speed liquid jet. O' Keefe et al. [4] presented a development of the Bow- den and Brunton technique for the production of high peed liquid jets by using a projectile with a velocity f 1.77 km/s. They described the motion of a water olumn through a tapered section by applying the 1- D, unsteady equations of compressible fluid flow. Fur- her analysis of the jet nozzle flow was mathematically resented in 1973 by Ryhming [5]. His model focused mainly on one-dimensional, incompressible flow. Gen- rally, when the liquid packaged in the nozzle cavity is mpacted by the high speed projectile, shock wave prop- gation and reflection in the liquid are always found. However, wave interactions were not considered in ei- her O' Keefe or Ryhming's study. Therefore, in 1977 esser [6] presented the basic mechanics of supersonic et generation by using the theory of guided acoustic waves. This used the fundamentals of shock propaga- on in the liquid to provide an estimation of the liquid et velocity created by the IDM. Accordingly, Shi's ex- eriment [7]demonstrated that the liquid shock reflec- on process took place in terms of the pressure history n the nozzle cavity and described the effects of shock eflection on the liquid jet characteristics. Recently, Pi- nthong et al. [8] presented a comprehensive one di- mensional model to describe the driven jet generation rocess during the short time interval in which the pro- ectile travels in the nozzle cavity both outwards and on ebound. This model considered the motion of the ini- al impact driven liquid shock in a single stepped nozzle om which the strength of its reflected shock wave was alculated. The pressure of the compressed liquid down- ream of the shock system within the sac could then be alculated and as this is the final driver of the jet which merges from the stepped nozzle the velocity of the high peed liquid jet followed. Their model results showed ood agreement to the previous experimental results. Moreover, the multiple pulsing phenomena presented in revious experiments was adequately described. How- ever, Pianthong et al.'s model is difficult to apply to ther nozzle cavity geometries, for example conical or urved, because of the higher dimensional effects. This drawback was confirmed by Matthujak's work [9] show- ing the nozzle sac pressure history in which the second and third shock reflection was captured using a pres- sure transducer. It was found that the driving pressures measured from these experiments were much lower than those determined from the calculations.

Recently, Computational Fluid Dynamics (CFD) nu- merical methods have been employed to investigate these high speed liquid jet characteristics. In 2003, Pianthong et al. [10] reported the simulation of the shock wave structure in air ahead of the emerged jet using a sta- tionary "solid"(i.e. fixed boundary) jet model that had been shaped by the steady flow compressed air field. Zakrzewski et al. [11] improved Pianthong et al's work by using the species transport equation to predict the liquid jet transient development. This described numer- ically the interaction between the air and the high speed liquid jet. Although, in previous studies, CFD results agreed well with the experimental results, the nozzle flow characteristics were not included in the simula- tions. Thus, further work from these research groups [12] simulated the full shock wave propagation within the nozzle, on all the nozzle materials and the emerg- ing jet using AUTODTN-2DTM software. That analy- sis shows the shock propagating in the projectile, liquid sac and nozzle material but it seems that this tool can- not precisely predict the transient development of the high speed liquid jet.

In this study, the simulation of the generation pro- cess of the pulsed high speed liquid jet is presented using the CFD program (FLUENT). Nozzle cavities with different geometries, including a stepped nozzle, a conical nozzle and a bell shaped nozzle are used to contain liquid water or diesel fuel. The simulation ap- proach was validated by comparison with experimental results from the previous studies of Pianthong [13,14], Shi [7,15], and Mutujak [9]. The shock wave reflection inside the nozzle cavity during the jet generation pro- cess is captured by the simulation. This clarifies how the pressure buildup inside the nozzle occurs to develop the liquid jet and provides a greater overall understanding of high speed liquid jets and their generation process. Such information will also provide a useful fundamental background for future studies of high speed jet injection in all the relevant fields.

2 Supersonic liquid jet generated by impact driven method

The principle of generating a high speed liquid jet by using the Bowden and Brunton method [3] is sketched in Fig.1. By this method, liquid retained in the nozzle is impacted by a high velocity projectile. On the impact, a



'ig. 1 Generation of high speed liquid jet by the impact riven method (IDM)

igh speed liquid jet forms and is ejected from the nozle to the test chamber. In some previous studies such as he experiments of Shi [7,15], the projectile did not imact directly on the liquid but onto a piston which then rove the liquid into the test chamber. This is called he momentum exchange method (MEM). In this paer, the works of Painthong [13,14], Shi, Shi [7,15] and latthujak [9] are used as the main references for invesigation of a supersonic liquid jet obtained using varius experimental techniques. In the work of Pianthong, shadowgraph optical system with a high speed video amera assisted in capturing the shock waves in series, hile the pressure history during the jetting process as obtained by Shi and by Matthujak using pressure ansducers. The details of their apparatus have been escribed in their studies.

CFD modeling of the generation process of npact driven high speed liquid jets

1 Geometric model

etails of the nozzle geometries used in this study are iown in Fig. 2. Two types of nozzles which are a conil and a stepped nozzle with cavity volume of 4.68 cm^3 id 6.54 cm^3, respectively, are investigated. Moreover, ie nozzle cavity shapes in the studies of Shi [7,15] have so been simulated to investigate the effect of the different jet generation methods; direct impact (IDM) and omentum exchange (MEM). In this study, the comercial CFD package (FLUENT) was used. Geometri-l domains of both nozzle cavity sac shapes are shown Fig. 2 while a parabolic cavity, volume of 0.42 cm^3, is picted in Fig. 3 for the two different methods. From e mechanism of high speed jet generation as shown

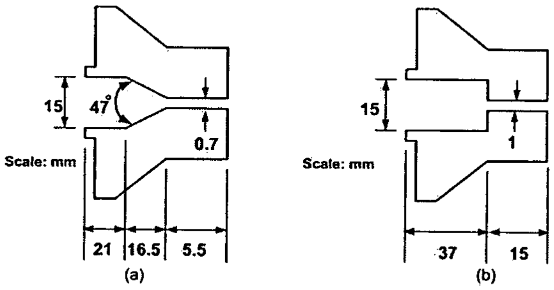


Fig. 2 Nozzle geometries (a) Conical nozzle and (b) stepped nozzle

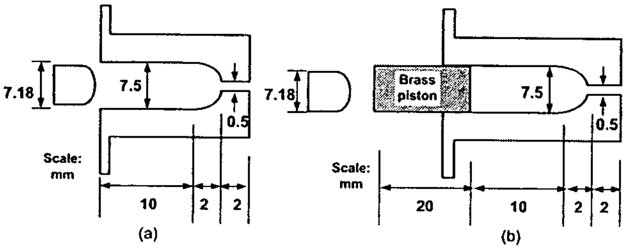


Fig. 3 Generation method (a) direct impact (IDM) and (b) momentum exchange (MEM)

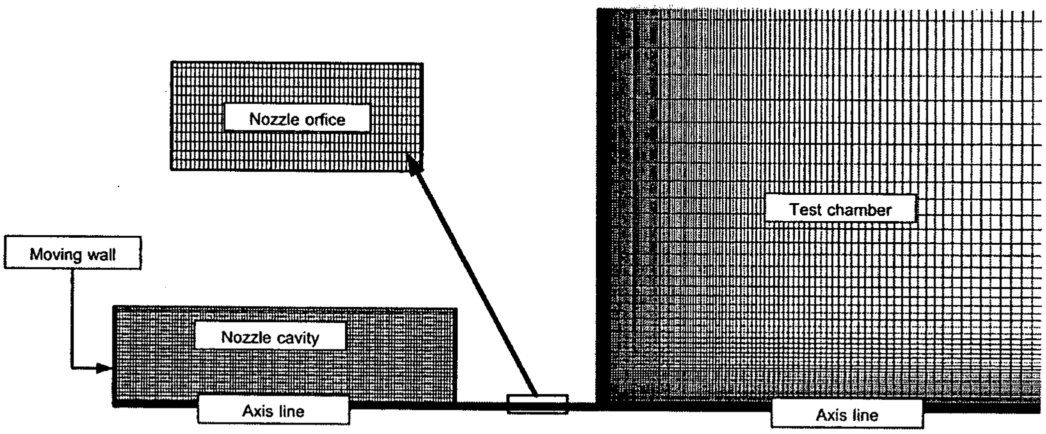
in Fig. 1, this setup can be modeled in a closed system domain with axi-symmetric geometry divided into a nozzle cavity zone and a test chamber zone as shown in Fig. 4, the test chamber zone being 50 mm height and 250 mm in width and meshed with 60,000 quadrilateral elements. This is fixed in all cases in this study; however the nozzle cavity region is varied, depending on the dimension and mesh size corresponding to the nozzle cavity lengths. In this transient zone, the interval size along the x-direction (dx) is fixed at 0.3 mm to provide the moving mesh for the projectile motion. In Fig. 4, the mesh is dense at the high shear layer area of interaction between the liquid jet and the quiescent air.

3.2 Projectile movement model

The motion of the projectile in the nozzle cavity is assumed to be that of a moving rigid wall from which the moving mesh of the nozzle cavity zone is constructed. The projectile velocity during the jet generation process after the impact can be computed from a simple force balance on the projectile front and the liquid package in the x-direction as

$$\int_{to}^{t} dV = \int_{to}^{t} (F(t)/m)dt \qquad (1)$$

where V is the projectile velocity, F is the driving force, and m is the mass of the projectile. The velocity



'ig. 4 Computational domain of axis-symmetric geometry of high speed liquid jet

.t any time t is calculated by using an explicit Euler ormula as

$$'_t = V_{t-\Delta t} + (F(t)/m)\Delta t \quad (2)$$

This formula is used to specify the motion of a movng wall (or projectile front wall) with linear velocities .t every time step (Δt) through the User Define Funcion (UDF) that is provided by the software. In the imulation, the masses used for the projectiles are 4.2 and 0.038 g which are similar to those in the studes of Pianthong [13,14], Matthujak [9], and Shi [7,15]. 'he force acting on the projectile in the x-direction simply the resistance force from the compressed liqid pressure, the friction force along the projectile wall eing neglected. For direct impact (IDM) of the proectile, initial velocities of 300 m/s, 700 m/s, and 414 1/s following the previous studies are set for the initial iovement of the wall. The velocity in the simulation of ie MEM can be computed from conservation of moientum in the system of the projectile and brass piston. 'he atmospheric pressure and ambient temperature are et as the initial condition in the domain. At times from ie projectile momentum remaining too high, the proectile can impact the nozzle trap (or front wall). In is situation, the projectile will release the excess moientum into the nozzle material bringing its velocity zero before it rebounds by the reaction force of the ompressed liquid. In addition, the calculation process completed when the projectile arrives at the entry oint. Because of the very different pressure across the vo phase zones, a pressure fluctuation can sometimes e induced by the high speed liquid jet generation; conequently, some of the liquid phase can be evaporated the gas phase by cavitation. Therefore, this pheomenon needs to be considered when the local pressure lower than the vapor pressure of the liquid. The full cavitation models presented by Singhal et al. [16] and Fluent user's guide [17] are applied to specify the vapor pressure and cavitation rate in the liquid and air flow. This assumption might not be completely accurate because the liquid must evaporate to its vapor gas, instead of air but it is acceptable. However, the properties of the liquid vapor in these studies and moist air are comparable.

3.3 Liquid properties model

At the initial condition, the two fluid phases were divided into a liquid water phase in the nozzle cavity and an air phase in the test chamber. The air density is simply specified by using the ideal gas formula to cope with the compressible flow field. Furthermore, in the nozzle cavity, it is much more complicated to specify the water as a compressible liquid. In this study, it can be modified by using a formula that includes the instantaneous liquid density (eq.(3)) and sound speed (eq.(4)) [18]. In the formula, the variables P and ρ are the liquid pressure and density respectively and the constant B is the bulk modulus of elasticity of the liquid. Subscripts 0 and 1 denote the respective quantity at the initial and current time level. In addition, the density and the sound speed are related to the corresponding time dependent liquid pressure. Liquids used in this study and their properties are listed in Table 1.

$$\rho_1 = \frac{\rho_o}{1.0 - (P_1 - P_0)/B} \quad (3)$$

$$a_1 = \frac{1 - (P_1 - P_o) \times \sqrt{B}}{B \times \rho_o} \quad (4)$$

ł.4 Solver modeling

[he CFD commercial code (FLUENT) is used as the ool to simulate the dynamic characteristics of the jet ;eneration process. A mixture model with velocity slip vas used for specifying the properties of the mixture vithin the multiphase flow. In the unsteady flow soution, the time step sized (Δt) of 0.1 microseconds vas set; therefore, results from each calculation can e recorded. The turbulence model is the standard k-ϵ nodel with segregate solver for the non-linear equaions.

Validation of CFD simulation

'his section presents the validation of the dynamic charcteristics of the jet generation process by comparing esults in this study with previous work. Water and iesel liquid jet characteristics, expressed as average ve-)cities defined as the jet penetration distance divided y jet emerging time, are shown in Fig. 5. These jets riven by a projectile having a velocity of 300 m/s, merge from the conical nozzle with geometry as shown ı Fig. 2(a). The average velocities calculated by the 'FD method are compared with the experimental reılts of Pianthong [13,14]. It is observed that the avrage jet velocity trends are only slightly different. In ıe first 30 microseconds, the experimental and CFD ɜsults are very close, however, in the following stage, ıe CFD simulation gives higher average velocities than ıose from experiments. Also, the calculated results ıow that the water jet velocity is higher than that of ıe diesel after 30 microseconds but not before, because ıe bulk modulus of elasticity of the water is greater ıan that of the diesel. For a larger bulk modulus, the ıange in fluid volume creates a greater internal presıre build-up with increase of the external pressure on ıe fluid, In addition, the diesel jet may form a more ;omized droplet cloud because of its lower surface tenon making it more easily dispersed into the air when starts to breakup.

In Fig. 6, the absolute static pressure histories inside ıe nozzle obtained from the experiments of Matthujak] are compared with the CFD simulation. In this case,

ıble 1 Properties of liquids

ـiquid	Bulk modulus (GPa)	Vapor pressure (Pa)	CP (J/(kg.K)	Surface tension (N/m)
Water	2.24	3169	4182	0.0717
Diesel	1.6	1378	1850	0.0244

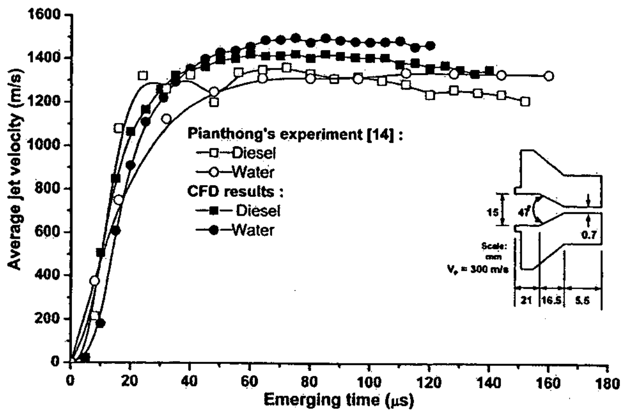


Fig. 5 Average jet velocities

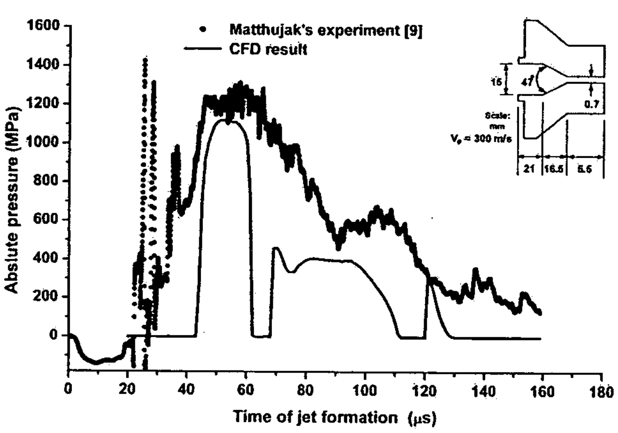
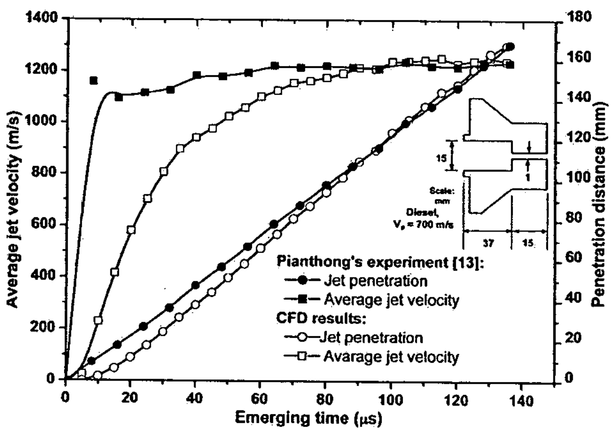


Fig. 6 Pressure history inside the nozzle cavity

the water is retained in a nozzle cavity with geometry as shown in Fig. 2(a) and driven by a projectile with velocity of 300 m/s. From Fig. 6, it was found that there were three peak pressures which were created by multiple water shock reflection during the jet generation process. However, the additional pressure fluctuations corresponding to the shock waves released from the nozzle body itself during the initial stage (20 microseconds) could not be captured by the CFD simulation, because that addition has not yet been considered in the CFD modeling. The value of the peak pressures obtained from the CFD are 1.1, 0.4 and 0.3 GPa while corresponding values from the experiment are 1.24, 0.6 and 0.27 GPa. That is, the results from the CFD are fairly similar to the experimental results with the exception that some peak pressures occur at different periods (i.e. the peak pressure of 2nd and 3rd pulse), the experiment giving a delay, of around 20 - 30 microseconds compared to the CFD. Because of the simple cavitation model employed in the CFD, the actual super cavita-



'ig. 7 Jet velocity and penetration distance

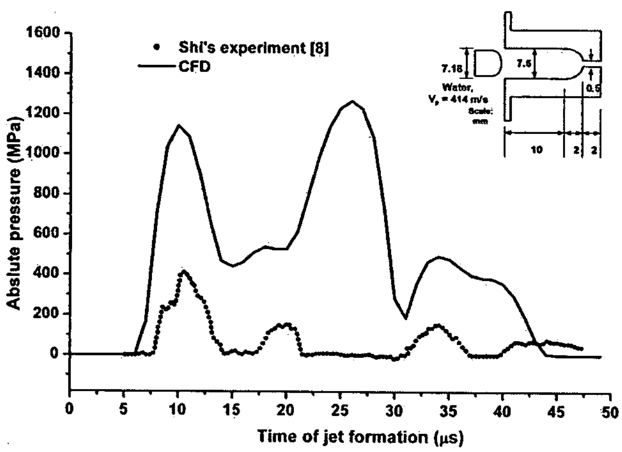


Fig. 8 Pressure history from direct impact of the projectile

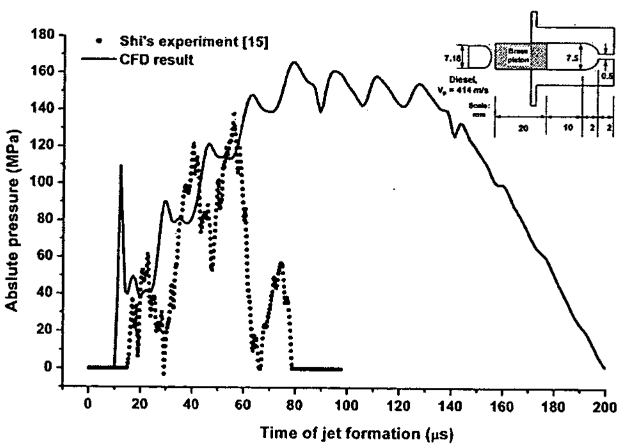


Fig. 9 Pressure history with a driven brass piston

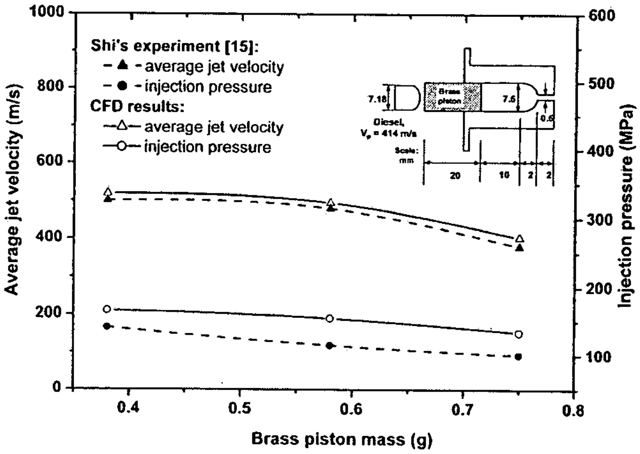


Fig. 10 Injection pressure and jet velocity with a driven brass piston

ion process occurring during the jet generation process ıside the nozzle cavity was not fully taken into account ı the CFD model.

Fig. 7 shows the characteristics of a high speed diesel ɜt in terms of velocity and penetration distance emerg- ıg from a stepped nozzle of geometry as shown in Fig. (b). This jet was generated by the impact of a projec- :le with velocity of 700 m/s. The experimental results f Painthong's study in 2003 [13], are compared with ıe simulation results. The average overall jet velocity f both results is quite similar, about 1100 m/s, even ıough in the simulation the jet accelerated more slowly ı the initial stage. From the penetration, the simula- on indicates that the high speed liquid jet might take ınger to accelerate than indicated by the experiments ır the first few microseconds; this being because in the ҁperiment, it is not possible to capture sufficient data oints in that time period. At 0 to 60 microseconds, the ⁄erage jet velocity from the experiment is higher than ıat from the calculation. This is because, in the exper- ıent, the shock propagation in the container material :sulting from the projectile impact may be released .to the liquid at an early stage, a phenomenon not ınsidered in the simulation.

Moreover, the CFD results agree in many respects ith the experimental results from Shi's studies [7,15] . which two impact methods, the IDM and the MEM, ·e used for generating the high speed liquid jet. For ɔth methods, the liquid was impacted by a projec- le of mass 0.384 g with velocity 414 m/s . The pres- ıres inside the nozzle cavity and the average jet veloci- ɜs which were achieved from the CFD simulation and ıese experiments are shown in Fig. 8-10. For the)M, the maximum driving pressures found from the .lculation and experiments are significantly different. owever, the pressure history trend determined from he pressure peaks agrees well during the jet generation ›rocess as shown in Fig. 8. Because the CFD model is , closed domain in which all boundary conditions set s walls, the leaking of liquid through the tolerance beween the nozzle hole and the projectile was not specfied, and a higher injection pressure is therefore likely rom the calculated results. In support of this, in Shi's xperiment, a large tolerance between the projectile and ozzle cavity was used this being, around 0.3 mm (proectile and nozzle cavity diameters being respectively .18 mm and 7.8 mm). Also, the efficiency of the imact of the experimental apparatus creates uncertainty 1 the experimental results. However, this only slightly ffects the reflected shock motion in the liquid during he process as confirmed by the close timing of the exerimental and simulated peaks as shown in Fig. 8.

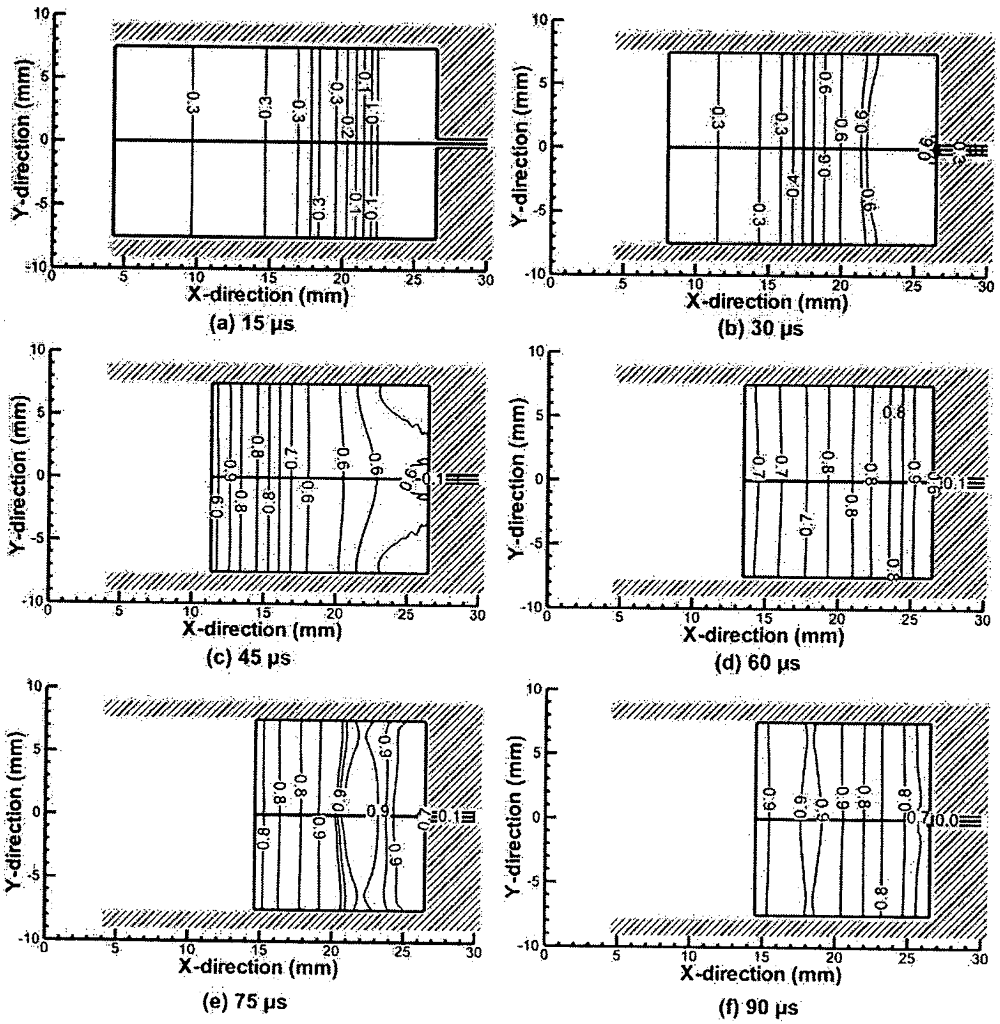
For the MEM in which the projectile impacts on le brass piston instead of the liquid, the maximum ijection pressures of around 140 MPa from Shi's exeriment are reasonably comparable with the simulaon at 160 MPa as shown in Fig. 9. With this method, le physical domain is similar to the closed domain of le simulation. Also, the CFD shows good agreement ith the experimental results as shown in Fig. 10 when le average jet velocity and maximum injection presire are related to the piston mass, Fig. 10 showing lat, when the piston mass is increased, the injection ressure and jet velocity decrease. This is due to the omentum conservation giving a slower piston movement. However, the duration of the jet generation pross from the CFD calculation is twice that from the exeriment. This is because the simulation assumes that le projectile always attaches to the piston and they len move together along nozzle cavity. In actuality, in le experiment, the projectile may bounce off the piston iring the process giving a duration of the momentum ansfer process that is significantly shorter and less fective. Moreover, in the experiment, a friction force ill also occur between the piston and the nozzle cavity all that will reduce the pressure rise in the liquid.

Dynamic characteristics of jet generation :ocess

ıis section presents the dynamic characteristics of the ; generation process resulting from the CFD simution. Illustrations of the pressure profiles inside the ızzle cavity are presented and described. The behavr of the liquid shock reflection in the nozzle cavity s the most significant effect on the high speed liquid ; characteristics. Therefore, this study shows the inantaneous pressure profile inside the nozzle, as shown Fig. 11 and 12 where the profile can be seen in a sequential time series. These figures show the instantaneous static pressures corresponding to shock waves in the diesel columns contained within both a stepped and a conical nozzle cavity respectively having volume and projectile velocities of 4.68 cm^3 and 300 m/s identically.

For the stepped nozzle of Fig. 11, it is shown that the pressure buildup near the nozzle shoulder occurs three times corresponding to the number of jet pulses. At an early stage (15 microseconds), Fig. 11(a), the static pressure near the projectile front is around 0.3 GPa which extends some distance in the x-direction before reducing further away. At 30 microseconds, the pressure builds up slightly first near the nozzle exit shoulder due to the wave reflection. When the pressure has accumulated adequately to more than the maximum pressure in the earlier stage, a reflected shock then travels back to the projectile face. At this stage, the high speed liquid jet forms. After the reflected shock wave reaches the projectile face, it reflects back again because of further pressure buildup. Therefore, this pressure is now added over the original shock reflection process and is a clear explanation of the formation of the multiple pulses within the high speed liquid jet. Pressure profiles inside a stepped - conical nozzle cavity is not similar to those in the plane stepped cavity in the pressure buildup process near the orifice entrance. Therefore, the value of the maximum pressure in both cases is significantly different. From Fig. 12, at an early stage (Fig. 12(a)), the high speed liquid flow resembles a converging wave through a nozzle, the conical cavity creating the highest pressure at the first pressure buildup pulse. Later, the pressure is greatly reduced by releasing the liquid into the air (as shown in Fig. 12(c)). This phenomena is unlike the stepped cavity where the maximum pressure always occurs at the second pulse of the liquid shock wave near the orifice entrance. This is because, in the stepped cavity, the pressure reduction of the first pulse is only slowly released and a high pressure is maintained near the nozzle to be built up again by the next pulse. However, in both cases, the third pressure pulse occurs similarly at the moment of projectile rebound.

Simulation results of the flow behavior within a parabolic cavity generated by the MEM and IDM are shown in Fig. 13 and Fig. 14 respectively. In the simulation, the projectile velocity is around 414 m/s and diesel liquid is used. It was found that the pressure created with the MEM is lower than with IDM. In addition, for the MEM, the pressure is slightly built up as the brass piston moves towards the nozzle and wave propagation cannot be observed. This is in accord with the results

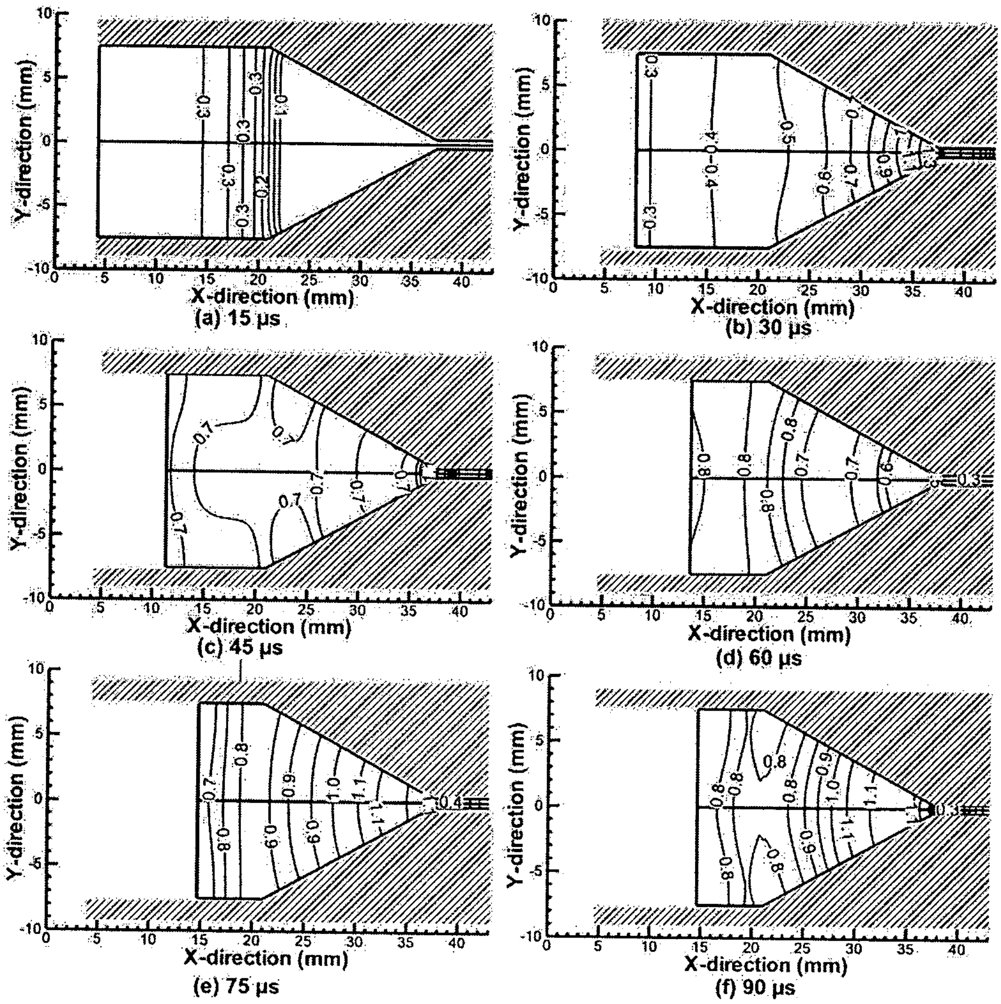


ig. 11 Pressure profile (GPa) of diesel inside a stepped nozzle

ıown in Fig. 9. Moreover, the traveling distance of ıe piston inside the nozzle is very short (around 1.8 m). Hence, some of the momentum of the projecle is still stored inside the piston body instead of beg completely transferred and the pressure inside the ızzle cavity is consequently low. However, the total omentum of the piston attached to the projectile is ıual to that in the direct impact method and would ıus require a longer duration of the jetting process to ansfer energy at low speed into the retained liquid. easonably, increasing the mass or length of the brass ston causes a reduction of the injection pressure and t velocity, as shown in Fig. 10. This is different to the ızzle flow behavior generated by the IDM. With such method, the pressure distribution during the process n be observed. In addition, the complex wave propation was also captured by the CFD simulation as shown in Fig. 14. Some of the momentum of the projectile was directly released into the liquid in a much shorter time; therefore, extremely high pressures and a wave propagation system are produced. The injection pressure is increased by the wave reflection inside the nozzle cavity. This is similar to the stepped nozzle as described in the previous section. However, values of injection pressure and pressure profile are different and depend on nozzle shape and geometry as shown in Fig. 11, 12 and 14.

6 Concluding remarks

In this study, high speed liquid jets have been numerically simulated by using the impact driven method which is from the interaction of a high velocity projectile on a liquid package contained in a nozzle cavity.



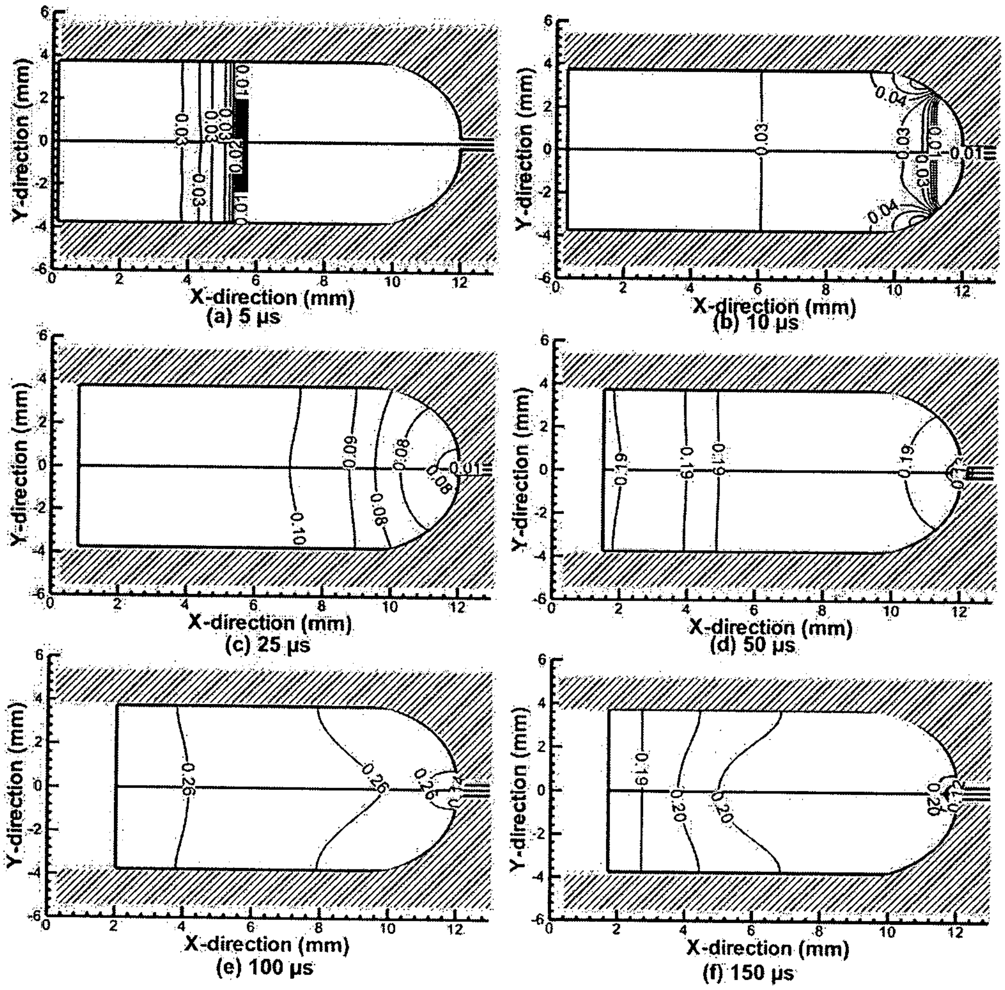
'ig. 12 Pressure profile (GPa) of diesel inside a conical nozzle

. Computational Fluid Dynamics (CFD) technique is mployed for simulation of the jet generation process by ıe IDM method within a closed domain for the first me. The two fluid model consisting of liquid and air ın be successfully calculated. Both direct impact and ıomentum exchange methods have been explored. The FD results show good agreement to previous experi-ıental results. It was found that, for a conical cavity, ıe first reflection peak produces the highest pressure, hile in the case of a stepped nozzle, the highest pres-ıre peak occurs at the second reflection. Moreover, the omentum exchange rate between projectile and com-·essed liquid during the jet generation process depends gnificantly on the nozzle cavity geometry more the ›lume of retained liquid. The direct impact method ·oduced higher driving pressures for the final jet efflux r an identical projectile momentum. This is because the stored momentum in the combined projectile and piston is released more slowly.

Acknowledgements This work was granted by the Office of the Higher Education Commission and Thailand Research Fund (RTF), contract NO. RMU5180020, the National Research Council of Thailand (NRCT) through Ubon Ratchatani University Research Grant fiscal year 2007. Wirapan Seehanam was supported by CHE Ph.D. Scholarship.

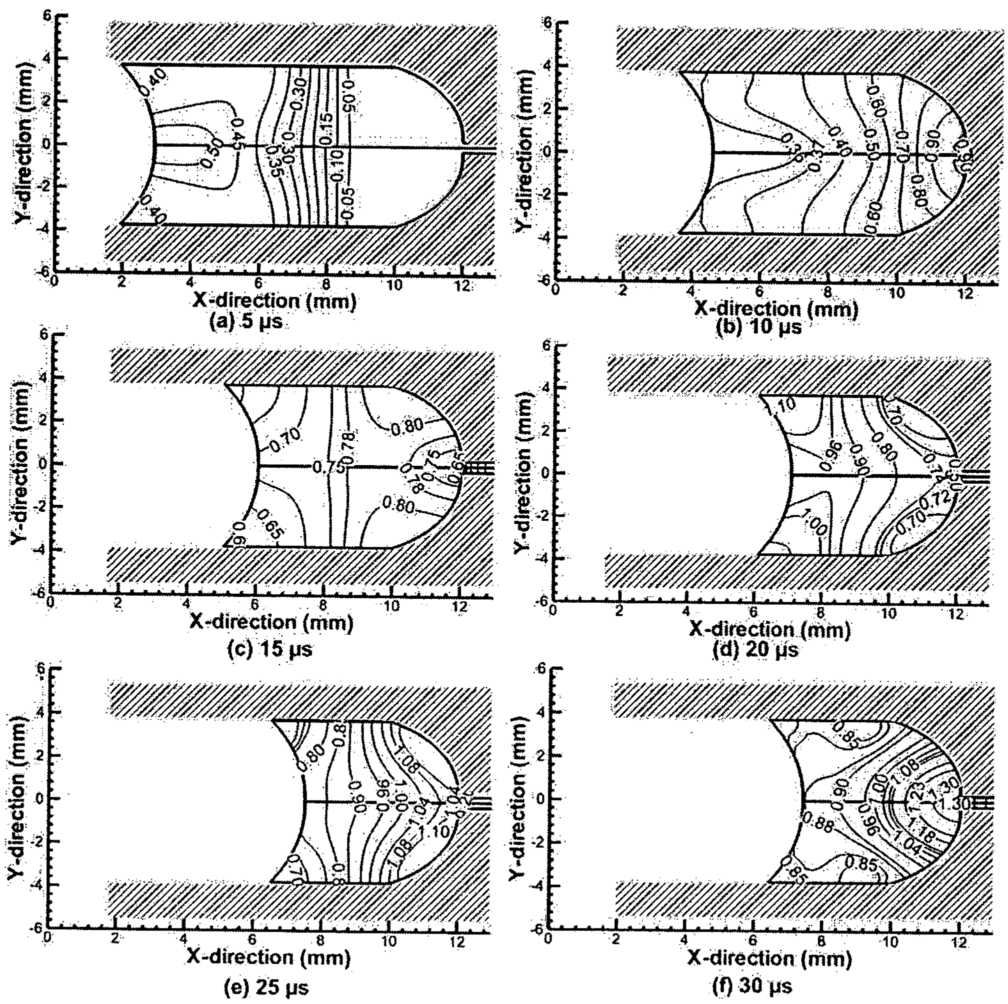
References

1. Giudice, E.L., Campbell, J.D.: Needle-free vaccine delivery. Adv Drug Deliver Rev. **58**, 68 - 89 (2005)
2. Baxter, J.S., Mitragotri, S.: Needle-free jet injections: dependence of jet penetration and dispersion in the skin on jet power.J Control Release. **97**, 527 - 535 (2009)



ig. **13** Pressure profile (GPa) created with the momentum exchange method

3. Bowden, F.P., Brunton, J.H.: Damage to solids by liquid impact at supersonic speed. Nature. **181**, 873 - 875 (1958)
4. Keefe, J.D.O., Wrinkle, W.W., Scully, C.N.: Supersonic liquid jet. Nature. **213**, 23 - 25 (1967)
5. Ryhm, I.R.:Analysis of unsteady incompressible jet nozle flow. Z. Angew. Math. Phys. **24**, 149 - 164 (1973)
6. Lesser, M.: Thirty years of liquid impact research: a tutorial review. Wear. **186-187**, 28 - 34 (1995)
7. Shi, H.H., Takayama, K.: Generation of high speed liquid jets by high speed impact of a projectile. JSME Int J., Ser. B. **38**, 181- 190 (1995)
8. Pianthong, K., Milton, B.E., Behnia, M.: Generation and shock characteristics of unsteady pulsed supersonic liquid jets. Atomization Sprays. **13**, 425 - 620 (2003)
9. Matthujak, A., Hossein, S.H.R., Takayama, K., Sun,M., Voinovich, P.: High speed jet formation by impact acceleration method. Shock Waves. **16**, 405 - 419 (2007)
10. Pianthong, K., Zakrzewski, S., Behnia, M., Milton, B.E.: Characteristics of impact driven supersonic liquid jets. Exp. Therm. Fluid. Sci.**27**, 589 - 598 (2003)
11. Zakrzewski, S., Milton, B.E., Pianthong, K., Behnia, M.: Supersonic liquid fuel jets injected into quiescent air. Int. J. Heat Fluid Flow. **25**, 833 - 840 (2004)
12. Pianthong, K., Matthujak, A., Takayama, K., Saito, T., Milton, B.: Visualization of supersonic liquid fuel jets. J. Flow Visual Image Process. **13**, 217 - 242 (2006)
13. Pianthong, K., Takayama, K., Milton, B., Behnia, M.: Multiple pulsed hypersonic liquid diesel fuel jets driven by projectile impact.Shock Waves. **14**, 73 - 82 (2005)
14. Pianthong, K., Matthujak, A., Takayama, K., Miton, B.E., Behnia, M.: Dynamic characteristics of pulsed supersonic fuel sprays. Shock Waves. **18**, 1 - 9 (2008)
15. Shi, H.H.: Study of Hypersonic Liquid Jets. Ph.D. Thesis,Tohoku University, Japan (1994)
16. Singhal, A.K., Athavale, M.M., Li, H., Jaing, Y.: Mathematical basic and validation of the full cavitation model. J. Fluids Eng. **124**, 617 - 624 (2002)
17. Fluent Inc.: Fluent 6.3 User Guides. New Hampshire, USA (2005)



ig. 14 Pressure profile (GPa) created with the direct impact method

18. Fluent Inc.: Fluent 6.3 UDF Manual. New Hampshire, USA (2005)

Elsevier Editorial System(tm) for Computers and Fluids
Manuscript Draft

Manuscript Number: CAF-D-11-00352

Title: Injection pressure and velocity of high speed liquid jets generated by the impact driven method

Article Type: Original Research Paper

Keywords: Computational Fluid Dynamics (CFD); High speed liquid jet; Compressible fluid; Impact driven method.

Corresponding Author: Dr. Kulachate Pianthong, Ph.D.

Corresponding Author's Institution: Ubon Ratchatani University

First Author: Wirapan Seehanam, Ph.D.Candidate

Order of Authors: Wirapan Seehanam, Ph.D.Candidate; Kulachate Pianthong, Ph.D.; Wuttichai Sittiwong, Ph.D.Candidate; Brian Milton, Ph.D.

Abstract: High speed liquid jets have been applied in many fields of engineering, science and medicine. Therefore, an investigation of their characteristics by modern and inexpensive methods will benefit all these areas. In this study, an extensive numerical investigation was carried out with the aim of understanding the dynamic behavior involved in generating pulsed, high speed diesel jets. To obtain a single pulse at such pressures, the Impact Driven Method (IDM) was used. A Computational Fluid Dynamics (CFD) technique has also been employed to simulate the jet generation process within a closed domain. The injection process requiring the interaction of two compressible fluids, liquid diesel of very low compressibility and air which is highly compressible, were successfully simulated by using a two phase flow mixture model with a moving mesh for the projectile motion. Validation and limitation of this numerical model with previous experimental results are presented and discussed. In addition, the main influencing factors which are the geometric sizes of the nozzle and the velocity and density of the projectile on the jet generation behavior are explored as well as the histories of the local pressure and velocity at the orifice entrance and exit respectively. This provides a better understanding of nozzle flow behavior under high speed jet injection and will provide useful fundamental knowledge for future studies of high speed injection and its related fields.

COVER LETTER FOR SUBMISSION OF MANUSCRIPT

Kulachate Pianthong (Ph.D.)
Department of Mechanical Engineering, Faculty of Engineering
Ubon Ratchathani University
85 Sathonlamark Road, Warinchamrab, Ubonratchathani, 34190, Thailand
Telephone: +66 45 353 309 Fax: +66 45 353 308 Mobile phone: +66 81 876 2506
E-mail: k.pianthong@ubu.ac.th

1st July 2011

Subject: **SUBMISSION OF A MANUSCRIPT FOR EVALUATION**

Dear Editor,

I am enclosing herewith a manuscript of a research article entitled *"Injection pressure and velocity of high speed liquid jets generated by the impact driven method"* for publication in ***"Computers & Fluids"*** for possible evaluation.

With the submission of this manuscript, I would like to undertake that;

- All authors of this research paper have directly participated in the planning, execution, or analysis of this study.
- All authors of this paper have read and approved the final version submitted.
- The content of this manuscript have not been copyrighted or published previously.
- The contents of this manuscript are not now under consideration for publication elsewhere.
- The contents of this manuscript will not be copyrighted, submitted, or published elsewhere, while acceptance by the journal is under consideration.
- There are no directly related manuscripts or abstracts, published or unpublished, by any authors of this paper.

Elsevier Editorial System(tm) for Computers and Fluids
Manuscript Draft

Manuscript Number: CAF-D-11-00352

Title: Injection pressure and velocity of high speed liquid jets generated by the impact driven method

Article Type: Original Research Paper

Keywords: Computational Fluid Dynamics (CFD); High speed liquid jet; Compressible fluid; Impact driven method.

Corresponding Author: Dr. Kulachate Pianthong, Ph.D.

Corresponding Author's Institution: Ubon Ratchatani University

First Author: Wirapan Seehanam, Ph.D.Candidate

Order of Authors: Wirapan Seehanam, Ph.D.Candidate; Kulachate Pianthong, Ph.D.; Wuttichai Sittiwong, Ph.D.Candidate; Brian Milton, Ph.D.

Abstract: High speed liquid jets have been applied in many fields of engineering, science and medicine. Therefore, an investigation of their characteristics by modern and inexpensive methods will benefit all these areas. In this study, an extensive numerical investigation was carried out with the aim of understanding the dynamic behavior involved in generating pulsed, high speed diesel jets. To obtain a single pulse at such pressures, the Impact Driven Method (IDM) was used. A Computational Fluid Dynamics (CFD) technique has also been employed to simulate the jet generation process within a closed domain. The injection process requiring the interaction of two compressible fluids, liquid diesel of very low compressibility and air which is highly compressible, were successfully simulated by using a two phase flow mixture model with a moving mesh for the projectile motion. Validation and limitation of this numerical model with previous experimental results are presented and discussed. In addition, the main influencing factors which are the geometric sizes of the nozzle and the velocity and density of the projectile on the jet generation behavior are explored as well as the histories of the local pressure and velocity at the orifice entrance and exit respectively. This provides a better understanding of nozzle flow behavior under high speed jet injection and will provide useful fundamental knowledge for future studies of high speed injection and its related fields.

For the Editors, I would like to disclose the following information and significant findings about this research paper:

In this study, a Computational Fluid Dynamics (CFD) technique has been employed for the simulation within a closed domain of the jet generation process by the IDM method. The CFD results show reasonable agreement to previous experimental results. The generation process of the pulsed high speed liquid jet and relationship between local injection pressure and jet velocity at the entrance and the exit of a nozzle orifice respectively are described. Moreover, the effects of velocities and densities of the projectile were investigated and are now clearly understood. It was found that the average jet velocity and the injection pressure had higher values with the impact of a projectile at either a higher velocity or with a greater density. From the simulation results, the pressure fluctuations inside the nozzle cavity were shown to have a close association to the liquid jet formation.

Sincerely yours,

Kulachate Pianthong
Corresponding author

Highlights

> We successfully employed CFD technique to describe the jet generation process by the IDM. >CFD results show good agreement to previous experimental results. >The multiple pulse behaviors in the nozzle are proven. >The pressure fluctuations inside the nozzle obviously dictate the jet characteristics.

Injection pressure and velocity of high speed liquid jets generated by the impact driven method

Wirapan Seehanam[1], Kulachate Pianthong,[1,*], Wuttichai Sittiwong[1], Brian Milton[2]

[1]Department of Mechanical Engineering, Faculty of Engineering, Ubon Ratchathani University, Ubonratchatani 34190, Thailand

[2]School of Mechanical and Manufacturing Engineering, University of New South Wales, Sydney, 2052, Australia

*Corresponding author

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

Phone: +66-4535-3309

Fax: +66-4535-3308

Email: K.Pianthong@ubu.ac.th

Abstract

High speed liquid jets have been applied in many fields of engineering, science and medicine. Therefore, an investigation of their characteristics by modern and inexpensive methods will benefit all these areas. In this study, an extensive numerical investigation was carried out with the aim of understanding the dynamic behavior involved in generating pulsed, high speed diesel jets. To obtain a single pulse at such pressures, the Impact Driven Method (IDM) was used. A Computational Fluid Dynamics (CFD) technique has also been employed to simulate the jet generation process within a closed domain. The injection process requiring the interaction of two compressible fluids, liquid diesel of very low compressibility and air which is highly compressible, were successfully simulated by using a two phase flow mixture model with a moving mesh for the projectile motion. Validation and limitation of this numerical model with previous experimental results are presented and discussed. In addition, the main influencing factors which are the geometric sizes of the nozzle and the velocity and density of the projectile on the jet generation behavior are explored as well as the histories of the local pressure and velocity at the orifice entrance and exit respectively. This provides a better understanding of nozzle flow behavior under high speed jet injection and will provide useful fundamental knowledge for future studies of high speed injection and its related fields.

Keywords: Computational Fluid Dynamics (CFD), High speed liquid jet, Compressible fluid, Impact driven method

1. Introduction

High speed liquid jets have been widely used for appropriate applications in many fields of engineering technology such as cutting technology, medical engineering, automotive and combustion technology. While some applications use a steady flow, many require pulsed jets either as single events or with a high repetition rate. Consequently, a fundamental knowledge of high speed liquid jets is essential for appropriate applications to these technologies. Over a number of years, researchers have studied the characteristics of such jets, especially the influence of the main factors associated with the relevant fields. Normally, such high speed liquid jets can be created from a highly compressed liquid flowing through the nozzle orifice. This high pressure can be obtained by two different methods. Either the work output of a mechanical device can be used directly or it can supply a high pressure reservoir. Alternatively, for single event applications, the momentum of a solid object impacting on the liquid can be used to build-up the liquid pressure. In all cases, the behavior of the injection pressure is affected by the upstream processes. These can be directly applied pressure from the operation of a mechanical device (e.g. the motion of a piston in a pump), the transient process following the opening of a valve from a high pressure reservoir or the momentum transfer from the projectile to the fluid. All in different ways modify the characteristics of the ensuing jet.

When a mechanical device such as a high pressure pump or intensifier is used in the commercial injection systems for combustion in modern diesel engines, the high pressure liquid is held in a tank (reservoir) before release into the nozzle. This method can also be used in water jet cutting technology. In the latter where the jets are continuous, the injection pressure is amplified from the high pressure pump in conjunction with an intensifier and the main factors influencing the cutting characteristics are the specific energy, width of cut, depth of kerf, jet velocity, and impact pressure [6, 4]. Moreover, the work of Jou [5] showed that the injection pressure mainly affects the formation of undulations on the specimens because it changed the magnitude of the impact force. But this injection pressure was not a factor affecting the jet pulsation frequency, it being more influenced by the nozzle orifice design. In Park and Heister's numerical work [9], a numerical method was applied to study the primary instability on viscous high speed liquid jets, it was found that the boundary layer thickness affected by the orifice design was the dominant factor affecting the wave growth near the nozzle exit. For the improvement of cutting efficiency, a jet entraining other matter such as solid particles and liquid droplets was produced. However, in the work of Sanada et al. [13], by using a high-speed steam-droplet jet, harsh erosion on the specimen surface was always found. Therefore, purified water has been preferred.

In fuel injection processes in diesel engines, fuel jets are intermittent and require a high pressure upstream of the nozzle that is repeated rapidly, each event being for a very short time period. It is well known that the combustion efficiency depends mainly on the injection pressure and the nozzle geometry. Therefore, many researchers have attempted to present the relationship between these factors and the jet characteristics such as its velocity and penetration. Many of these formulas were derived from constant speed operation and with a relatively low injection pressure characteristic of now out-of-date diesel engines and furthermore, did not consider nozzle flow during the injection. In 2009 Sezal et al. [15] investigated a nozzle flow with a high injection pressure of around 60 MPa by using Computational Fluid Dynamics. They found that the instantaneous maximum pressure peak could be three time higher than the prescribed pressure at the nozzle inlet due to shock reflection at the bottom of the nozzle sac. This causes an error in the predicted results achieved from the calculation when those formulas are applied to modern diesel engines in which the injection pressures have been continuously raised in recent years now being often in excess of 160 MPa. Moreover, their application to a high speed fuel (or any other liquid) jet obtained by using the impact method to build-up the liquid pressure may not be accurate.

For driving the flow by use of a solid object, (called the "driving object method"), the liquid contained in the nozzle cavity is driven by an object such as a piston which increases the liquid pressure by direct contact thereby accelerating the jet to a high efflux velocity. The retained liquid is accelerated by a piston, its own movement being generated from either a spring or high pressure gas expansion. The method can be applied to novel drug delivery devices (called "Needle free jet injection devices") that are used for providing a jet that penetrates the skin. In such an injection, because of shock waves propagating and reflecting within the nozzle cavity, an oscillating pressure occurs at the entrance of the nozzle orifice exit [1], with the average pressure being slightly decreased [14] to zero at the end of the injection process. Certainly, the jet characteristics depend mainly on the nozzle flow behavior caused by the variation of the nozzle cavity pressure. However, for this method, the relation between the pressure in the nozzle and the jet velocity has not been clearly presented in any previous work.

For a fundamental study of high speed liquid jets resulting from even higher pressures (i.e. ultra high pressures, exceeding 1 GPa), the jets can be obtained by using the approach called the "Impact Driven Method" (IDM) first presented by Bowden and Brunton [2] which is slightly different to the "driving object method". The liquid contained in the nozzle cavity is driven by the impact of a high speed solid projectile. In this method, shock waves from the impact directly raise the liquid pressure causing the jet to accelerate. Generally, like the driving object method, the nozzle flow behavior during the jet generation process directly affects the characteristics of the emerging high speed liquid jet. When the liquid contained within the nozzle cavity is impacted by the high speed projectile, shock wave propagation and reflection in the liquid are always found. O'Keefe et al. [8] presented an analysis of the Bowden and Brunton technique describing the motion of the liquid (which was water in their case) column through a tapered section by applying the 1-D, unsteady equations of compressible fluid flow. Recently in 2003, a new analysis of the jet nozzle flow was presented mathematically by K. Pianthong et al. [11]. They developed a model that described the driven jet generation process for a single impact during the short time interval in which the projectile continues to progress within a simple cylindrical stepped nozzle cavity causing shock propagation both outwards and on rebound. Their one-dimensional model results showed good agreement to the previous experimental results as well as providing an adequate description of the multiple pulsing phenomena found in their experiments. However, their model is difficult to apply to other nozzle cavity geometries, for example conical or curved types, because of the higher dimensional effects. This drawback was confirmed by Matthujak's experimental work [7] in which the conical nozzle sac pressure history for the second and third shock reflection was captured using a pressure transducer. It was found that the driving pressures measured from these experiments were much lower than those determined from the one-dimensional calculations. In addition, Pianthong et al. [10, 12] attempted to describe the relation between the pressure peaks and the pulsed jet that formed during the projectile impact period on the liquid inside the conical nozzle. However, due to experimental limitations, this relationship could not be fully clarified and the influence of such main factors as the nozzle geometry, the projectile density and the velocity of the projectile on the jet formation and nozzle flow was not investigated in their study.

In the present study, the effects of the major factors are explored, these being the dimensional sizes of the stepped nozzle, the velocity and density of the projectile on the jet generation behavior and the histories of the local pressure and velocity at the orifice entrance and exit respectively by using the CFD program (FLUENT). The simulation approach was first validated by comparison with the experimental results from previous studies of Mutujak et al. [7], Pianthong et al. [10,12], and Shi [16]. The generation process of the pulsed high speed liquid jet and the relationship between local injection pressure and jet velocity at the entrance and exit of a nozzle orifice are now described. This has clarified the details of how the pressure buildup inside the nozzle and the development of the liquid jet occurred and has provided a greater overall understanding of high speed liquid jets and their generation process. Such information will provide a useful fundamental knowledge for future studies of high speed jet injection in all the relevant fields.

2. Mechanism of IDM

The jet generation process considered here is the driving object method which is specified as the impact driven method (IDM) and the momentum exchange method (MEM) as shown in Fig. 1. With the IDM, first presented by Bowden-Brunton [2], the liquid retained in the nozzle is directly impacted by a high velocity projectile. On impact, a high speed liquid jet forms and is injected from the nozzle into the test chamber as shown in Fig. 1(a). With the other method called MEM, presented by Shi [16], the projectile impacts on a piston instead of directly on the liquid as shown in Fig. 1(b).

3. CFD modeling

3.1. Mixture model

A two-phase mixture model is employed to analyze the thermal and fluid dynamic behavior of the high speed liquid jet being considered. In the mixture model the fluid is considered to be a single fluid with two phases and the coupling between them is strong while each phase has its own velocity vector and there is a certain fraction of each phase within a given control volume. The unsteady state continuity equation for the mixture can be written as

$$\frac{\partial(\rho_m)}{\partial t}+\frac{\partial(\rho_m u_{m,i})}{\partial x_i}=0 \tag{1}$$

The momentum equation for the mixture can be obtained by summing the individual momentum equations of all phases. It can be expressed as

$$\frac{\partial(\rho_m u_{m,j})}{\partial t}+\frac{\partial}{\partial x_i}(\rho_m u_{m,i}u_{m,j})=-\frac{\partial p}{\partial x_j}+\frac{\partial}{\partial x_i}\left[\mu_m\left(\frac{\partial u_{m,i}}{\partial x_k}+\frac{\partial u_{m,k}}{\partial x_i}\right)\right]+F_j+\sum_{k=1}^{n}\frac{\partial}{\partial x_i}(\alpha_k\rho_k u_{dr,k,i}u_{dr,k,j}) \tag{2}$$

where i and j are coordinate index , F is a body force, and α_k is the volume fraction of phase k. In Eq. (1) and (2), μ_m is the viscosity of the mixture and ρ_m is the mixture density:

$$\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \tag{3}$$

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \tag{4}$$

In addition, u_m is the mass-averaged velocity or velocity of mixture and $u_{dr,k}$ is the drift velocity for the secondary phase k:

$$u_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k u_k}{\rho_m} \tag{5}$$

$$u_{dr,k} = u_k - u_m \tag{6}$$

The drift velocity (u_{dr}) is related to the relative velocity (slip velocity) (u_{pq}) and is expressed as follows

$$u_{dr,p} = u_{pq} - \sum_{k=1}^{n} c_k u_{qk} \tag{7}$$

where c_k is the mass fraction for any phase (k). The relative velocity is defined as the velocity of a secondary phase (p) relative to the velocity of the primary phase (q):

$$u_{pq} = u_p - u_q \tag{8}$$

Following FLUENT's user guide [3], the form of the relative velocity is given by:

$$u_{pq} = \frac{\rho_p d_p^2}{18\mu_q} \frac{(\rho_p - \rho_m)}{f_{drag}\rho_p} a \tag{9}$$

where τ_p is the particle relaxation time, d is the diameter of the particles of the secondary phase p. The default drag function f_{drag} is taken from

$$f_{drag} = \begin{cases} 1 + 0.15\,\mathrm{Re}^{0.687} & \mathrm{Re} \le 1000 \\ 0.0183\,\mathrm{Re} & \mathrm{Re} \ge 1000 \end{cases} \tag{10}$$

and the acceleration a is of the form

$$a = g - u_m \frac{\partial u_m}{\partial x_i} - \frac{\partial u_m}{\partial t} \tag{11}$$

The energy equation for the mixture takes the following form:

$$\sum_{k=1}^{n}\frac{\partial}{\partial t}(\alpha_k\rho_k E_k)+\sum_{k=1}^{n}(\rho_k E_k+p)\frac{\partial}{\partial x_i}\alpha_k u_{k,i}=\frac{\partial}{\partial x_i}(k_{eff}\frac{\partial T}{\partial x_i}) \tag{12}$$

The value of k_{eff}, the effective conductivity computed from:

$$k_{eff}=\Sigma\alpha_k(k_k+k_t) \tag{13}$$

where k_k is the thermal conductivity and k_t is the turbulent thermal conductivity defined according to the turbulence model. Because the standard $k-\varepsilon$ turbulence model is used in this study, k_t can be computed from:

$$k_t=\frac{c_p\mu_t}{\Pr_t} \tag{14}$$

where c_p is the specific heat, $\Pr_t$ is the turbulent Prandlt number (Pr = 0.85 in this case). The turbulent viscosity, μ_t, is computed by combining turbulence kinetic energy k and rate of dissipation ε, as follows:

$$\mu_t=\rho_m C_\mu\frac{k^2}{\varepsilon} \tag{15}$$

where C_μ is the constant in the turbulence model. To fulfill the governing equations of high speed liquid jet dynamic fields in turbulence phenomena, the standard $k-\varepsilon$ turbulence model is considered in this study. The $k-\varepsilon$ model introduces two new equations, one for the turbulent kinetic energy (k) and the other for the rate of dissipation (ε). These two equations can be expressed as the transport equations:

$$\frac{\partial}{\partial t}(\rho_m k)+\frac{\partial}{\partial x_i}(\rho_m k u_{m,i})=\frac{\partial}{\partial x_i}\left[\left(\rho_m C_\mu\frac{k^2}{\varepsilon}\right)\frac{\partial k}{\partial x_i}\right]+\rho_m C_\mu\frac{k^2}{\varepsilon}\left(\frac{\partial u_{m,i}}{\partial x_l}+\frac{\partial u_{m,j}}{\partial x_i}\right)\frac{\partial u_{m,i}}{\partial x_l}-\rho\varepsilon-2\rho\varepsilon M_t^2 \tag{16}$$

$$\frac{\partial}{\partial t}(\rho_m\varepsilon)+\frac{\partial}{\partial x_i}(\rho_m\varepsilon u_{m,i})=\frac{\partial}{\partial x_i}\left[\left(\rho_m C_\mu\frac{k^2}{\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right]+C_{1\varepsilon}C_\mu\rho_m k\left(\frac{\partial u_{m,i}}{\partial x_l}+\frac{\partial u_{m,l}}{\partial x_i}\right)\frac{\partial u_{m,i}}{\partial x_l}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{17}$$

The model constant $C_{1\varepsilon}$, $C_{2\varepsilon}$, and C_μ in Eqs. (16) and (17) are assigned the following default values:

$$C_{1\varepsilon}=1.44,\ C_{2\varepsilon}=1.92,\ C_\mu=0.09$$

M_t is the turbulent Mach number that accounts for the compressibility effect in the k–e model.

The volume fraction equation for the secondary phases can be obtained from:

$$\frac{\partial}{\partial t}\left(\alpha_p\rho_p\right)+\frac{\partial}{\partial x_i}\left(\alpha_p\rho_p u_{m,i}\right)=-\frac{\partial}{\partial x_i}\left(\alpha_p\rho_p u_{dr,p,i}\right)+\sum_{q=1}^{n}\left(\dot{m}_{qp}-\dot{m}_{pq}\right) \tag{18}$$

where $\dot{m}_{qp}$ denotes the mass transfer from phase q to phase p, and $\dot{m}_{pq}$, from p to q.

3.2. Cavitation model

Because of the very high pressure gradient across the two phase zones, a pressure fluctuation can sometimes be induced by the high speed liquid jet generation. Consequently, some of the liquid phase is evaporated to the gas phase by cavitation. The full cavitation models presented by the Fluent user's guide [3] and Singhal et al. [17] are applied to specify the vapor pressure and cavitation rate in the liquid and air flow. This assumption might not be entirely accurate because the liquid must evaporate to its own vapor gas, instead of to air but it is acceptable. However, the properties of the liquid vapor in these studies and moist air are reasonably comparable.

The full cavitation model accounts for all first-order effects (i.e. phase change, bubble dynamics, turbulent pressure fluctuations and non-condensable gases). The fluid is assumed to be a mixture of liquid, vapor and non-condensable gases. The fluid density is a function of the vapor mass fraction f which is computed by solving the mass and momentum conservation equations coupled with a vapor transport equation:

$$\frac{\partial}{\partial t}\left(\rho_m f\right)+\frac{\partial}{\partial x_i}\left(\rho u_{v,i} f\right)=\frac{\partial}{\partial x_i}\left(\gamma\frac{\partial f}{\partial x_i}\right)+R_e+R_c \tag{19}$$

where ρ_m is the mixture density, u_v is the velocity vector of the vapor phase, γ is the effective exchange coefficient. The vapor generation rate R_e and condensation rate R_c can be computed from the following:

when $p<p_v$

$$R_e=C_e\frac{\sqrt{k}}{\sigma}\rho_l\rho_v\sqrt{\frac{2\left(p_v-p\right)}{3p_l}}\left(1-f_v-f_g\right) \tag{20}$$

when $p>p_v$

$$R_c=C_c\frac{\sqrt{k}}{\sigma}\rho_l\rho_l\sqrt{\frac{2\left(p-p_v\right)}{3p_l}}f_v \tag{21}$$

Here C_e and C_c are empirical constants with values of 0.02 and 0.01 respectively, k is the local turbulence kinetic energy, σ is the surface tension coefficient of the liquid. Subscripts l, v and g denote the liquid, vapor, and non-condensable gas phases respectively.

3.3. Domain and geometrical

The stepped nozzle geometry used in this study is shown in Fig. 2. The two main geometric parts are the cavity (sac) and the orifice tube. Variables L_o, L_c and D_n are cavity length, orifice length and orifice diameter respectively. The geometrical domain and grid construction are shown in Fig. 3. From the mechanism of high speed jet generation as shown in Fig. 1, this can be modeled in a closed domain with an axi-symmetric geometry divided into a nozzle cavity zone and a test chamber zone. The test chamber space, being 50 mm height and 250 mm width, was meshed with nearly 60,000 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 volume. In this transient zone, the interval size along the x-direction (dx) is fixed at 0.3 mm to provide a moving mesh for projectile motion. The mesh was more densely created at the region of high shear interaction between the high speed liquid jet and the quiescent air.

3.4. Model of projectile movement and liquid properties

The movement of the projectile in the nozzle cavity can be represented by the motion of a rigid wall and hence a moving mesh was constructed for the nozzle cavity zone. A dynamic layering method was employed to update the mesh volume in the deforming cavity defined at the wall boundaries. The projectile velocity which equals the wall movement during the jet generation process after the impact can be computed from a simple force balance between the projectile front and the liquid package in the x-direction. It is assumed that the force acting on the projectile in the x-direction is simply the resistive force caused by the compressed liquid pressure with the relatively small friction force along the projectile wall being neglected. Thus, the velocity at any time t can be calculated using an explicit Euler formula as

$$V_t = V_{t-\Delta t} + (F(t)/m)\Delta t \tag{22}$$

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 the moving wall (or projectile front wall) with the linear velocities within every time step (dt) through the User Define Function (UDF) provided by the software.

At the initial condition, the two fluid phases were divided into a liquid phase in the nozzle cavity and an air phase in the test chamber. The air density was simply specified by using the ideal gas formula to cope with the compressible flow field in the simulation. However, within the nozzle cavity, specifying the liquid as compressible is much more complicated. In this study, its compressibility has been incorporated by using formulas that include the instantaneous liquid density (Eq. (23)) and the sound speed (Eq. (24)).

$$\rho_1 = \frac{\rho_o}{\left[1.0-(P_1-P_0)/B\right]} \tag{23}$$

$$a_1 = \frac{1-(P_1-P_0)}{B} \times \frac{\sqrt{B}}{\rho_o} \qquad (24)$$

In these equations, the variables P and ρ are the liquid pressure and density respectively and the constant B is the bulk modulus of elasticity of the liquid. Subscript 0 and 1 denote the respective values at the initial and current time levels. In addition, the density and the sound speed are time dependent corresponding to the liquid pressure. Properties of the liquids used in this study are shown in Table 1.

3.5. Algorithm

The CFD commercial code (FLUENT) was employed to solve the present problem. The governing equations were solved by the control volume approach. A first order upwind discretization scheme was used for momentum, turbulence kinetic energy, volume fraction of vapor and turbulence energy dissipation rate while a body-force weighted scheme was used for pressure. The SIMPLE algorithm was employed for the pressure-velocity coupling. In the unsteady flow solution, the time step size was set (dt) equal to 0.1 μs and the results from each calculation were recorded.

4. Validation of CFD simulation

This section presents the validation of the pressure inside the nozzle cavities and the jet velocities obtained by comparing results from the simulation with previous experimental work. The comparisons are shown in Tables 2 to 4. The peak pressures and the time intervals between them compared the absolute static pressure histories inside the nozzle obtained from the CFD simulation with the experiments of Matthujak et al. [7]. In these, the liquid (water) retained in the cavity of a conical nozzle was driven by a projectile with a velocity of 0.3 km/s. Multiple pressure peaks caused by the shock reflection during the jet generation process were found from both the CFD and the experiment. The calculated pressures are lower than the corresponding experimental ones for the first two peaks while the third CFD peak has a fairly similar although slightly higher value as shown in Table 2. A noticeable difference in the results from the calculation is that the first duration is considerably shorter than that shown by the experiment while the second is longer. However, the total time for the two increments is quite close. This is because both the friction and the leakage of liquid through the tolerance gap between the nozzle hole and the projectile were not specified in the calculation and this has a most noticeable effect on the first shock wave reflection cycle. Also, the efficiency of the impact during the experiment creates some uncertainty in the experimental results.

In the work of Pianthong et al. [10,12], the high speed jet flow was visualized by using a shadowgraph optical system with a high speed video camera which captured an image series of the jet emerging from a nozzle, This was used to measure the jet's average velocity for each camera frame step. Moreover, in this work, the effect of type of liquid was investigated as well as the nozzle geometry on the jet flow behavior. To validate the simulation approach of the present research, these were reproduced using the commercial code. The maximum jet velocities are compared in Table 3. It can be seen for the conical nozzle that the CFD simulation gives velocities about 5 to 10% higher than those from experiments. Little difference occurs between water, diesel and kerosene in the experiments although the last of these is the lowest but the CFD shows that the jet velocity of the water is higher than that of both the liquid fuels. This is because the bulk modulus of elasticity of the water is greater than that of the liquid fuels. For a larger bulk modulus, the change in fluid volume creates a greater internal pressure build-up as the applied impact force on the fluid increases. In addition, the fuel jet may form a more highly atomized droplet cloud as it emerges from the nozzle because its lower surface tension makes it disperse more easily into the air when it starts to breakup thus affecting the measurements downstream of the nozzle exit. For the stepped nozzle, the calculated results show that the diesel jet velocity is essentially the same in both experiment and calculation. In relation to the effect of cavity geometry, even when lower projectile velocities are used, the nozzle with the conical cavity gives a higher maximum velocity than that obtained with the stepped cavity.

An alternative impact method is the Momentum Exchange Method (MEM) in which the projectile impacts on a brass piston instead of directly on the liquid as shown in Fig. 1(b). In the simulation of this case, for each value of the piston mass, the maximum jet velocity and the injection pressure obtained from Shi's experiment [16] are also compared reasonably with the simulation, the CFD giving slightly higher values of both. Also, the CFD shows good agreement with other experimental results which are that when the piston mass is increased as shown in Table 4, the injection pressure and jet velocity decrease. This is due to momentum conservation giving a slower piston velocity.

The simulation shows some differences from experiment but these are explainable and are mostly due to the physical limitation of including such things as clearances, friction and impact efficiency. It is noted that the extreme pressures and the very short time scales also provide limitations related to both experimental accuracy and some secondary phenomena not being included in the simulation. However, the results are sufficiently similar for the simulation to be used to assess many aspects of the jet generation process.

5. Results and discussions

This section presents the characteristic profiles of the diesel jet generation process resulting from the CFD simulation. Illustrations of the pressure and velocity profiles inside a nozzle cavity and the jet characteristics in the test chamber respectively are presented and discussed. Moreover, investigations are included on the effect of the main parameters which are the orifice diameter, cavity length, orifice length, projectile density and projectile velocity on the local histories of the static pressure at the orifice entrance and the liquid velocity at the orifice exit.

5.1. Profiles of pressure inside the cavity and the jet velocity

The behavior of the multiple liquid shock reflections in the nozzle cavity has a very significant effect on the high speed liquid jet characteristics. Therefore, this study has examined the instantaneous pressure profile inside the nozzle, as shown in Fig. 4 where the profile can be seen in a sequential time series. This figure show the instantaneous jet velocity profile corresponding to compression waves in the diesel columns contained within a stepped nozzle cavity. A cavity with a volume of 4.68 cm^3 and projectile with a velocity of 0.3 km/s are used. The contours illustrated show the direction of the compression wave movement and the pressure buildup by the impact corresponding to the jet pulses. At an early stage (20 μs), the compression waves in front of the projectile front travel in the x-direction towards the nozzle orifice. At 40 μs, the pressure builds up slightly, first near the nozzle exit shoulder due to the wave reflection. When the pressure has accumulated adequately to more than the maximum pressure of the earlier stage, a reflected wave then travels back to the projectile front bringing the injection static pressure at that location to around 0.68 GPa. This pressure is retained for some distance in the x-direction before reducing towards the nozzle. The first-pulse jet with velocity of 0.82 km/s forms during this period. Then, by 80 μs, the pressure further builds up slightly beginning near the nozzle exit shoulder due to another wave reflection. As the pressure accumulates reaching a higher value than that at the earlier stage, a reflected shock travels back to the projectile face and a second jet pulse having a velocity of 0.98 km/s is created through the orifice as shown. This is because the pressure at the entrance to the nozzle which is that driving the jet is now higher than that of the original shock reflection process. As the projectile starts to rebound due to the high pressure at its face and its loss of momentum to the liquid, the pressure throughout the cavity starts to fall. This can be seen at times of 120 and 160 μs. At 120 μs, the 0.98 km/s jet velocity contour has started to detach from the nozzle due its lower driving pressure and by 160 μs has moved well away from the exit. Closer to the nozzle exit, the jet velocity has fallen firstly to 0.82 km/s and finally right at the exit to 0.76 km/s This provides a clear explanation of the formation of the multiple pulses observed in experiments within such high speed liquid jets.

5.2. Phase change of high speed jet liquid

Usually, the process of breaking up into fine droplets always occurs under a high velocity efflux. This causes a phase change from liquid droplet to gas phase by vaporization. The high speed liquid jet generated by the IDM can be clearly explored by the CFD simulation. Fig. 5 illustrates the phase change based on the variation of the volume fraction along the axis of the jet (or center line of the core jet), generated by a single projectile impact, at 80, 120, and 160 μs of the emerging time. It is found that the volume fraction plot shows pulsating behavior where the value of the fraction peak, being the dense cloud jets, corresponds to the pulse of the compression wave, as shown in Fig. 4, that is taking place inside the nozzle cavity during the jet generation. At the early 80 μs emerging time, the first and second jet are found. The second jet, which just emerges from the orifice and is very close to the nozzle exit, has volume fraction near to 1.0, i.e. it is still liquid. Later on, the diesel liquid jet atomizes and evaporates into fine droplets and vapor due to the high shear force resulting from the interaction between the high speed liquid jet and the quiescent air. Consequently, the volume fraction of the jet is significantly decreased with further travelling distances and longer emerging times. In addition, when the volume fraction at 120 μs and 160 μs are tracked, as in Fig. 5, it is found that the first jet disappears since the diesel phase completely changes into vapor during this time, while the second and third jets contain a much lower volume fraction.

5.3. Velocity and pressure at orifice entrance and exit

This section of the study is to explore the dynamic behavior of the spray parameters and the relationship between the flow phenomena inside the nozzle and the jet formation at the nozzle exit. Therefore, the injection pressure and velocity histories at the orifice entrance and orifice exit respectively have been highlighted and observed. In the simulation, the geometry of the stepped nozzle has not been altered, i.e. it is 1 mm in nozzle orifice diameter, 15 mm in cavity diameter, 26.5 mm in cavity length, 7.857 mm in nozzle orifice length. Moreover, the projectile has generally been kept fixed in velocity, 0.3 km/s and weight, 4.2 g. However, these have occasionally been varied to investigate their effects on the jet generation process and flow behavior.

5.3.1. Effect of orifice diameter

It is well known that orifice diameter is the main factor dominating the characteristics of a fluid spray. Thus, in this section, the numerical investigation of such a factor is presented. Orifice diameters of 0.5, 1.0, 1.5, and 2 mm were used with the other parameters held constant. In this case, the diesel was retained in the stepped nozzle cavity as detailed in the previous section and driven by a projectile with a velocity of 0.3 km/s. The results in terms of the local histories at the orifice entrance and nozzle exit during the jet generation process have been observed and are shown in Fig. 6. It can be seen that the pressure build-up is produced by multiple shock reflections within the diesel in the nozzle cavity. This causes jet velocity pulses at the nozzle exit, with four peaks of pressure and velocity that can be correspond in the observations. The maximum values are at the second pulse. The peak velocity is delayed by over five μs from the corresponding pressure peak in each case. This trend depends very slightly on the orifice diameter and is consistent for multiple pulses over 100 μs after the jet first emerges. Although, the nozzles with the smallest orifice can produce a higher static pressure, this seemingly does not translate to a higher velocity at the second and third pressure peak. This is most probably because intensive atomization and cavitation during the jet flow occur more readily with the higher liquid pressure coupled with its flow through the smaller orifice. Moreover, the range of orifice diameters used in this work is not large enough to have a significant influence on the jet flow.

5.3.2. Effect of cavity length

The length of the cavity was changed in the simulation to investigate the effect of the frequency of the shock reflections and the cavity length on the nozzle driving pressure and the jet flow. The lengths used were 10, 20, 30, and 40 mm with the 1 mm orifice diameter and 0.3 km/s projectile velocity being retained as previously. Fig. 7 shows the variation of the local histories for these different configurations. At the third peak, the shortest cavity length of 10 mm gave the highest local jet velocity of around 1.3 km/s with a peak pressure of around 1.3 GPa. As the cavity was lengthened, the peak pressure decreased as did the corresponding velocity but the process duration extended. Surprisingly, even with the shortest cavity of 10 mm, the projectile did not strike the downstream shoulder of the cavity during the process. Moreover, four peak pressures and their corresponding pulsed jets were observed in all cases. This is because an identical projectile momentum for each case is transferred to the retained liquid, the longer nozzle cavity resulting in an extended process duration. That is, the shock pattern has to travel a greater distance which may be alternatively viewed as the cavity with a larger volume having a greater mass to absorb the high kinetic energy from the projectile. Therefore, the process must require a longer duration to reduce the impact energy.

5.3.3. Effect of orifice length

Because the orifice flow phenomena directly affect the spray characteristics, this work looked at the influence of the geometric dimensions of the orifice, this being the length to diameter ratio. In this simulation, the 1 mm orifice was retained but with lengths ranging from 0.5 to 20 mm. The other parameters, cavity volume, orifice diameter and projectile velocity were held at the previous values. The results as shown in Fig. 8 illustrate how these lengths dominate the jet generation and its characteristics. With an extended orifice length, the initial creation of the pulsed jet was slightly delayed although the pressure peak is not. Velocities at this stage varied very little between the cases although the longer orifice delayed the jet formation. The highest jet velocity was obtained at the second pulse in all cases where again its values did not vary much but the extended delay with the longer orifice remained. Here, the pressure peak became only very slightly higher as the orifice was lengthened. For all cases, after the first and second peaks, the pressure peaks trended downwards and their values were reasonably independent of the orifice length. Between the first and second pressure peaks, the shorter orifices of lengths 5 and 10 mm had pressure histories in the range from about 20 – 80 μs that varied noticeably from those of the longer orifices .but at later times, all pressure histories came close to merging. This is because of the emergence of the first jet pulse which influences the quiescent liquid at the orifice entrance causing it to develop a high velocity.

5.3.4. Effect of projectile density

In the jet generation process using the IDM, the mass of the projectile is the main factor that supplies the kinetic energy that builds up the injection pressure. However, previous experimental studies have not considered the influence of such a factor in their discussion. In this study, it has been investigated and revealed for the first time. For a constant projectile volume, the mass varies directly with the projectile density which in practice could be altered by the use of various materials. Projectiles having densities of 400, 800, 1200, and 1600 kg/m^3 were used in the simulation with a 26.5 mm long cavity and a 7.857 mm orifice length. From the results as shown in Fig. 9, it is found that increasing the projectile density does produce a higher jet velocity and injection pressure while also extending the duration of the jet generation. A maximum jet velocity of 1.2 km/s and a pressure of 0.9 GPa were found at the third peak of the curve, these being obtained with the projectile of the highest density, 1600 kg/m^3. In addition, the local jet velocity and pressure strongly depend on the projectile density. The high projectile density gives a high kinetic energy which results in a long duration that maximises the momentum transfer during the jet generation. This also produces the highest pressure and jet velocity for the same impact velocity.

5.3.5. Effect of projectile velocity

Because the characteristics of the jet created by using the IDM are directly dependent on the projectile momentum which can be obtained not only from the projectile density but also from the projectile velocity, many researchers have investigated the effect of the projectile velocity on the jet by using either an experiment apparatus or a mathematical model. Nevertheless, none of these thoroughly reveal how those parameters relate to the characteristics of the injected jet. This is because in experiments it is hard to obtain direct measurements of the jet characteristics occurring during the instantaneous generation process and also because accurately maintaining the velocity of the projectile in experiments is difficult. By using the CFD simulation in this study, the influence of the projectile velocity can be clearly explored and presented. A projectile having velocities ranging from 0.3 – 0.7 km/s and a 4.2 g mass was studied. The same nozzle geometry was used as in the previous section that examined the effect of projectile density. Fig. 10 shows the effect of the projectile velocity on the histories of the jet velocity at the nozzle exit and the pressure at the orifice entrance. It is found that both the velocity and the injection pressure rise to high values with the impact of the high velocity projectile. Four pulsed jets and pressure peaks occur and a delay time for the creation of the first jet from the first pressure peak of about 10 μs can be observed in all cases. For values of projectile velocities above 0.6 km/s, the projectile was found to strike the cavity shoulder resulting in a reduction of the jet generation duration as well as the pressure and velocity values after the second peak. Thus, the histories differed from those of the non-striking projectiles which have velocities up to 0.5 km/s. This is because the momentum of a projectile is not only transferred into the retained liquid but is also released into the nozzle material during the projectile impact on the container.

6. Concluding remarks

In this study, a Computational Fluid Dynamics (CFD) technique has been employed for the simulation within a closed domain of the jet generation process by the IDM method. The CFD results show reasonable agreement to previous experimental results. The generation process of the pulsed high speed liquid jet and relationship between local injection pressure and jet velocity at the entrance and the exit of a nozzle orifice respectively are described. Moreover, the effects of velocities and densities of the projectile were investigated and are now clearly understood. It was found that the average jet velocity and the injection pressure had higher values with the impact of a projectile at either a higher velocity or with a greater density. From the simulation results, the pressure fluctuations inside the nozzle cavity were shown to have a close association to the liquid jet formation.

7. Acknowledgments

This work was granted by the Office of the Higher Education Commission and Thailand Research Fund (TRF), contract No. RMU5180020, the National Research Council of Thailand (NRCT) through Ubon Ratchatani University Research Grant fiscal year 2007. Wirapan Seehanam was supported by CHE Ph.D. Scholarship.

8. References

[1] Baker AB, Sanders JE. Fluid mechanics analysis of a spring-loaded jet injector. IEEE Trans Biomed Eng 1999; 46(2): 235-242.

[2] Bowden FP, Brunton JH. The deformation of solids by liquid impact at supersonic speeds. Proc R Soc A 1961; 263: 433-450.

[3] FLUENT 6.3 User's guide. Fluent Inc., 2006.
[4] Harris HD, Mellor M. Cutting rock with water jets. Int J Rock Mech Min Sci Geomech Abstr 1974; 11: 343-358.
[5] Jou M. Analysis of the stability of water-jet cutting with linear theory. J Mater Process Technol 2000; 104: 17-20.
[6] Leach SJ, Walker GL, Smith AV, Farmer IW, Taylor G. Some aspects of rock cutting by high speed water jets. Philos Trans R Soc A 1966; 260: 295 -310.
[7] Matthujak A, Hossein HR, Takayama K, Sun M, Voinovich P. High speed jet formation by impact acceleration method. Shock Waves 2007; 16: 405 – 419
[8] O'Keefe JD, Wrinkle WW, Scully CN. Supersonic liquid jet. Nat 1967; 213: 23 - 25.
[9] Park H, Heister SD. A numerical study of primary instability on viscous high-speed jets. Comput Fluids 2006; 35: 1033-1045.
[10] Pianthong K, Matthujak A, Takayama K, Milton BE, Behnia M. Dynamic characteristics of pulsed supersonic fuel sprays. Shock Waves 2008; 18: 1 - 10
[11] Pianthong K, Milton BE, Behnia M. Generation and shock characteristics of unsteady pulsed supersonic liquid jets. Atomiz Sprays 2003; 13: 425-620
[12] Pianthong K, Takayama K, Milton B E and Behnia M. Multiple pulsed hypersonic liquid diesel fuel jets driven by projectile impact. Shock Waves 2005; 14: 73 - 82
[13] Sanada T, Watanabe M, Shirota M, Yamase M. Impact of high-speed steam-droplet spray on solid surface. Fluid Dyn Res 2008; 40: 627- 636.
[14] Schramm J, Mitragotri S. Transdermal drug delivery by jet injectors: energetics of jet formation and penetration Pharm Res 2002; 19(11): 1673-1879
[15] Sezal IH, Schmidt SJ, Schnerr GH, Thalhamer M, Frster M. Shock and wave dynamics in cavitating compressible liquid flows in injection nozzles. Shock Waves 2009; 19: 49-58.
[16] Shi HH. Study of hypersonic liquid jets. Ph.D. Thesis, Tohoku University, Sendai, Japan; 1975.
[17] Singhal AK, Athavale MM, Li H, Jaing Y. Mathematical basic and validation of the full cavitation model. J Fluids Eng 2002; 124: 617 - 624.

Figure

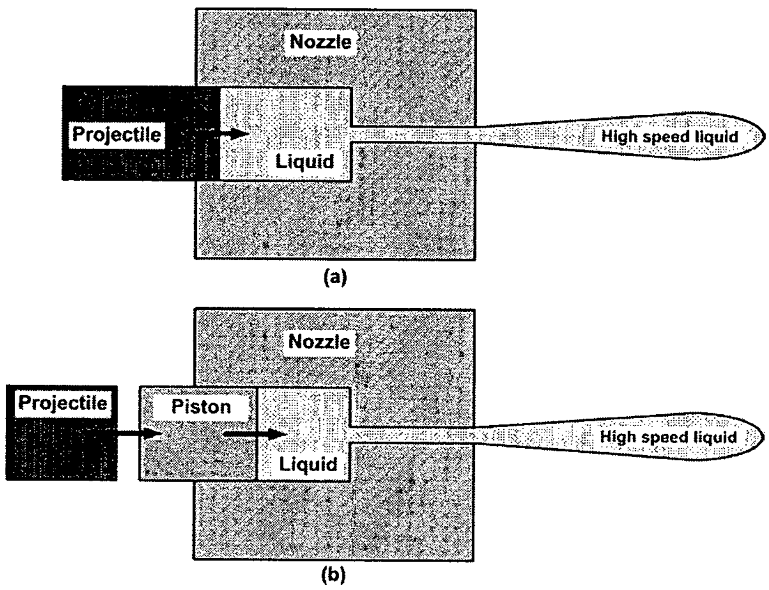


Fig. 1. Generation of a high speed liquid jet by (a) the impact driven method (IDM) and (b) the momentum exchange method (MEM)

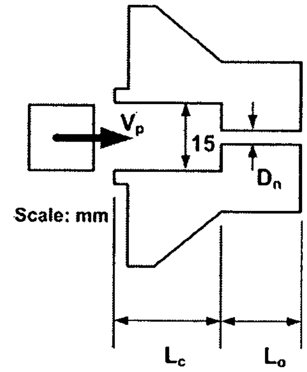


Fig. 2. Nozzle geometry

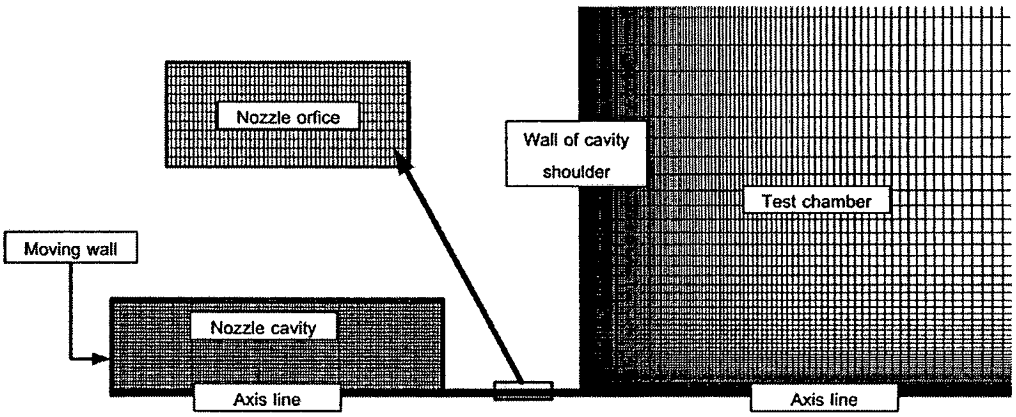


Fig. 3. Computational domain of the axi-symmetric geometry for high speed liquid jet simulation

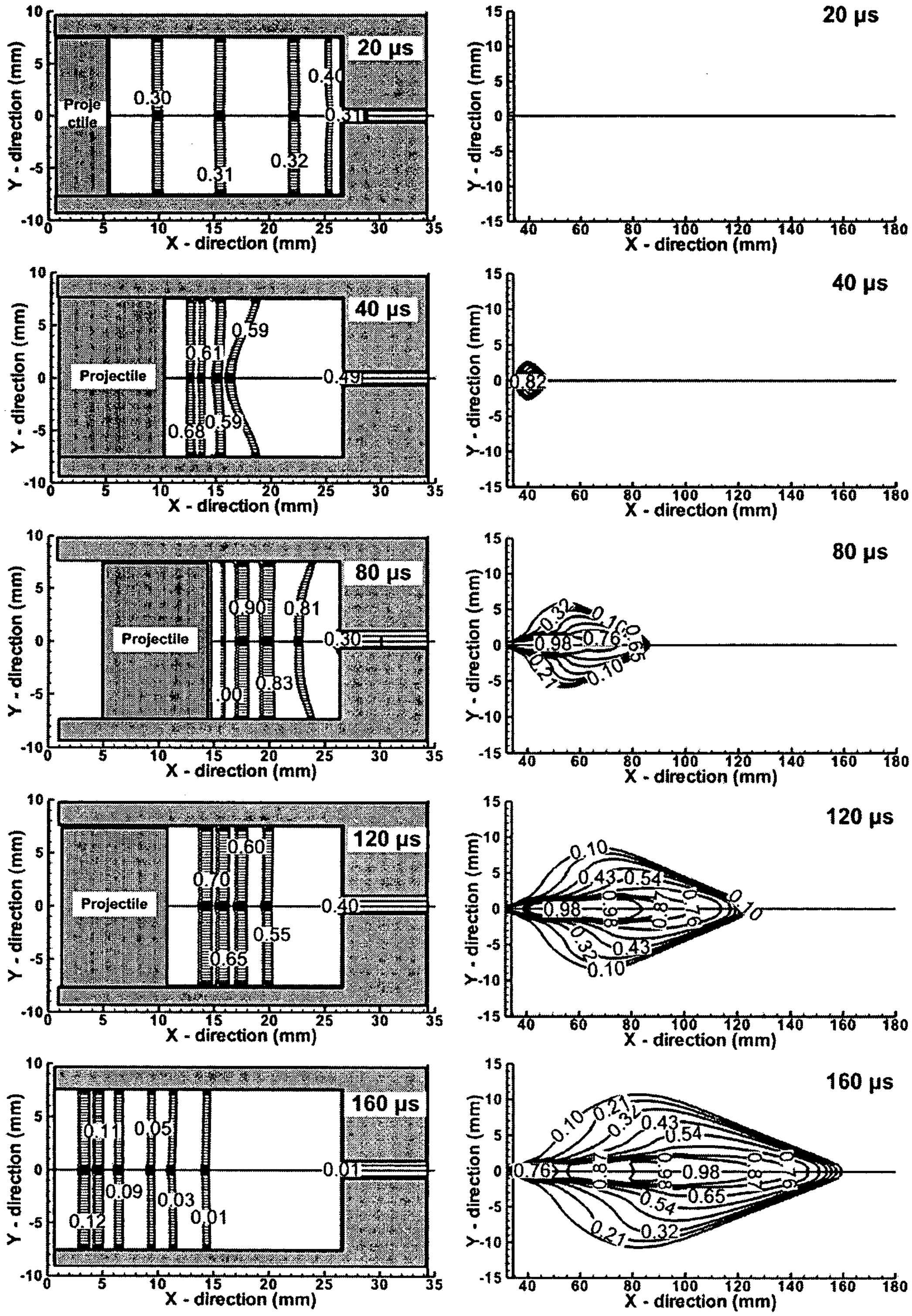


Fig. 4. Pressure (GPa) and jet velocity profile (km/s)

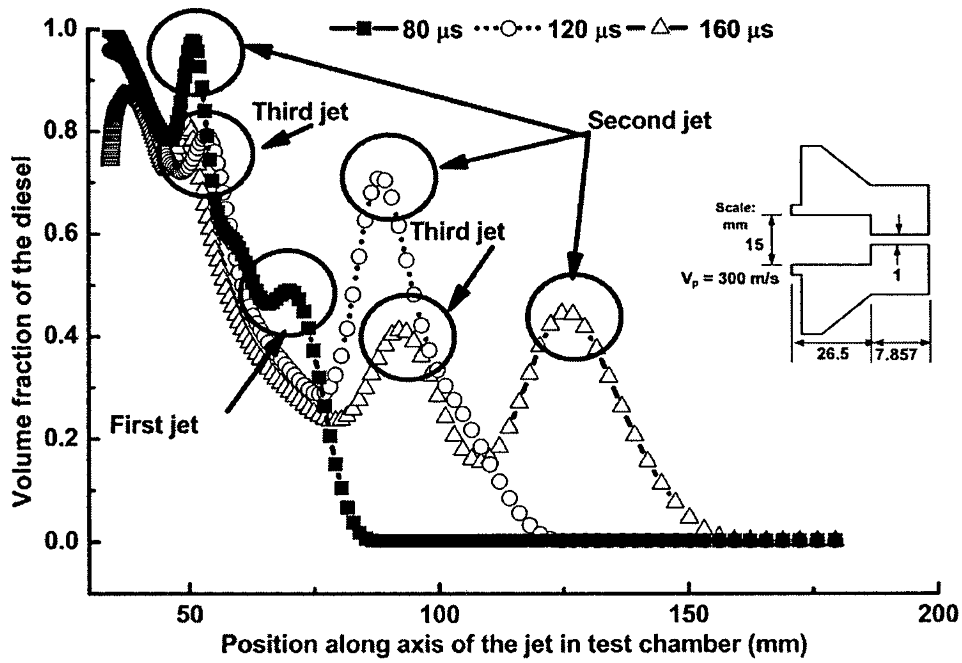


Fig. 5. Volume fraction of diesel along the jet axis

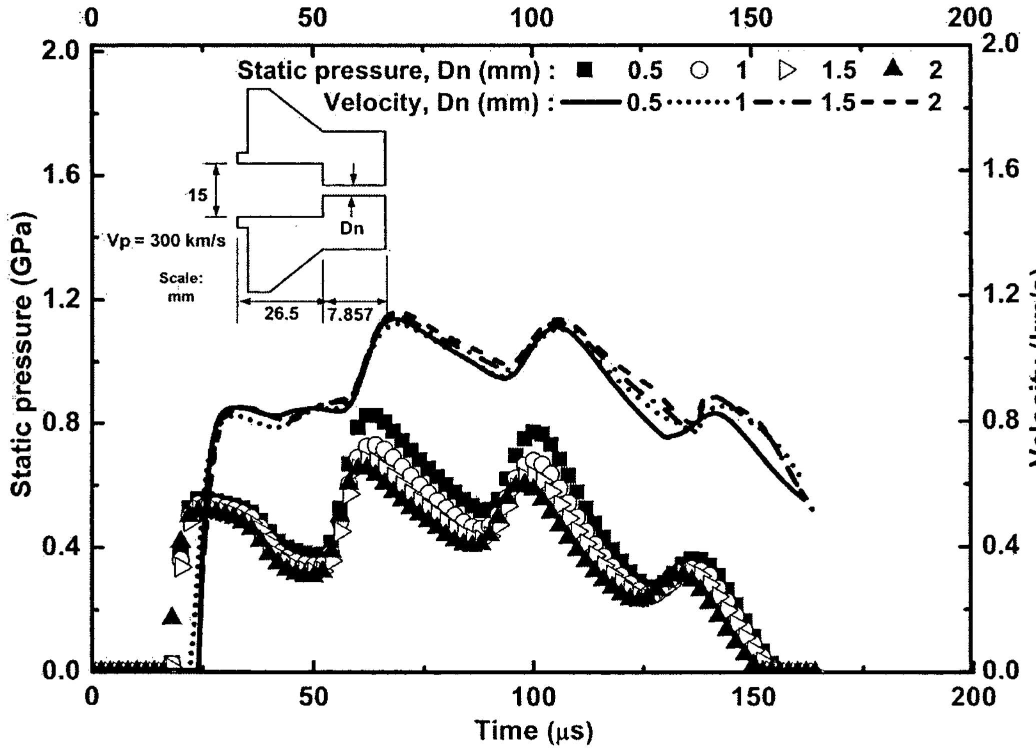


Fig. 6. Injection pressure and velocity with various orifice diameters

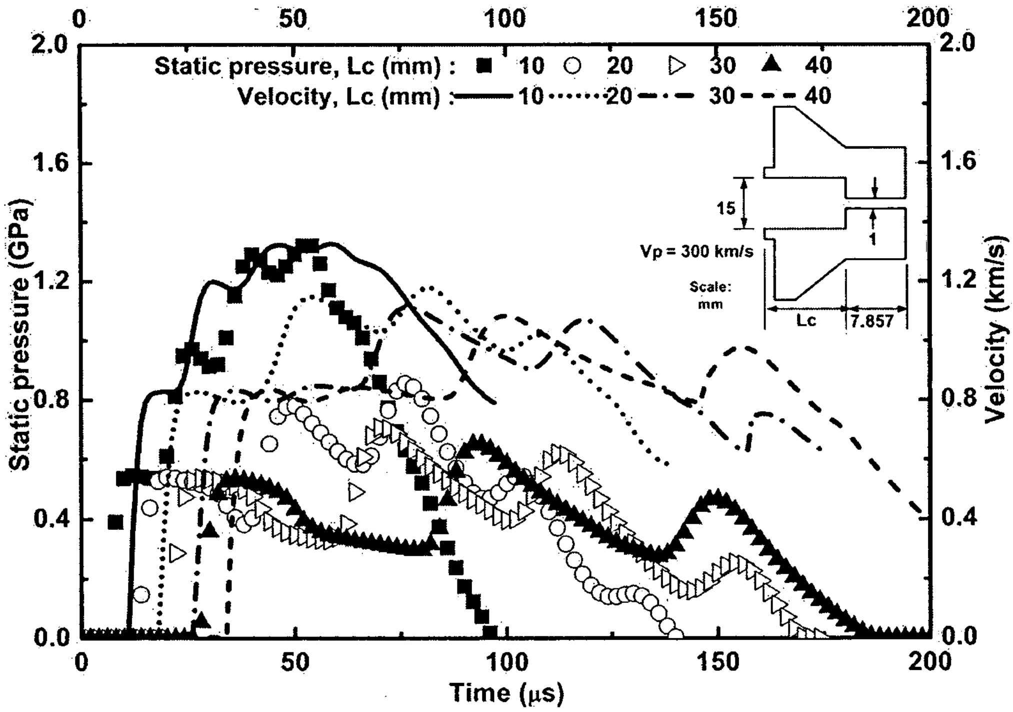


Fig. 7. Injection pressure and velocity with various cavity lengths

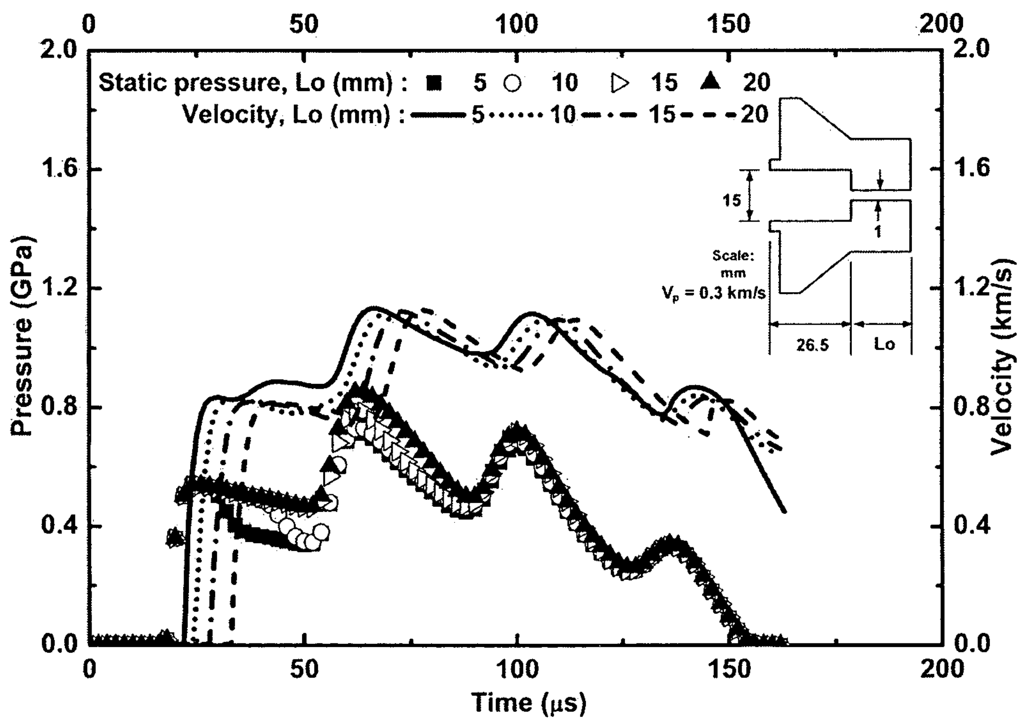


Fig. 8. Injection pressure and velocity with various orifice lengths

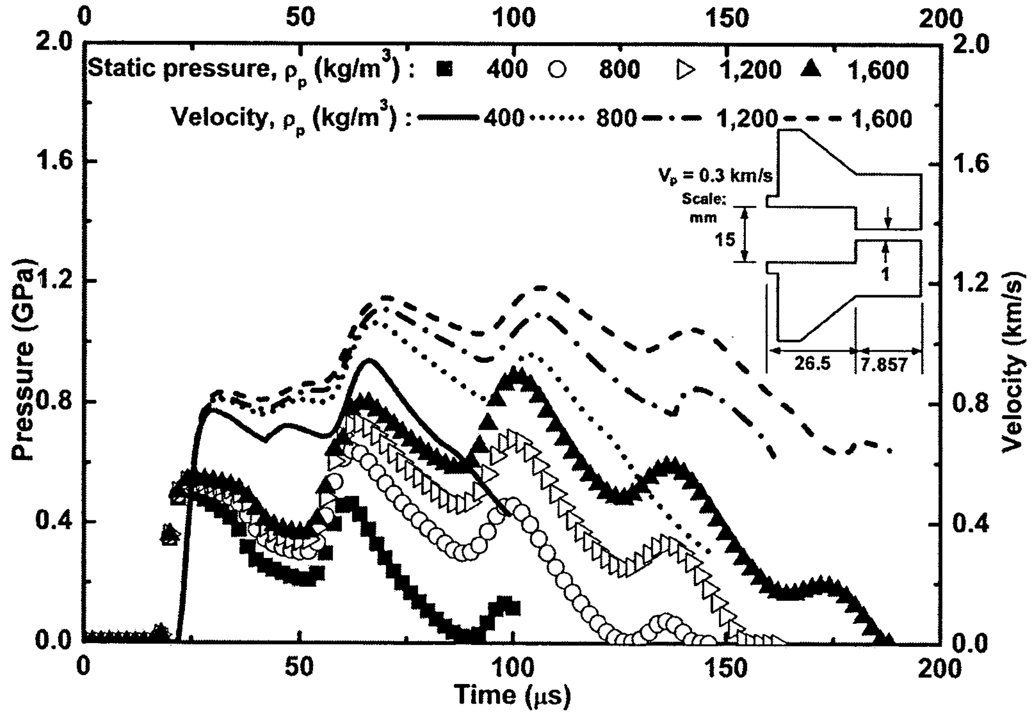


Fig. 9. Injection pressure and velocity with various projectile densities

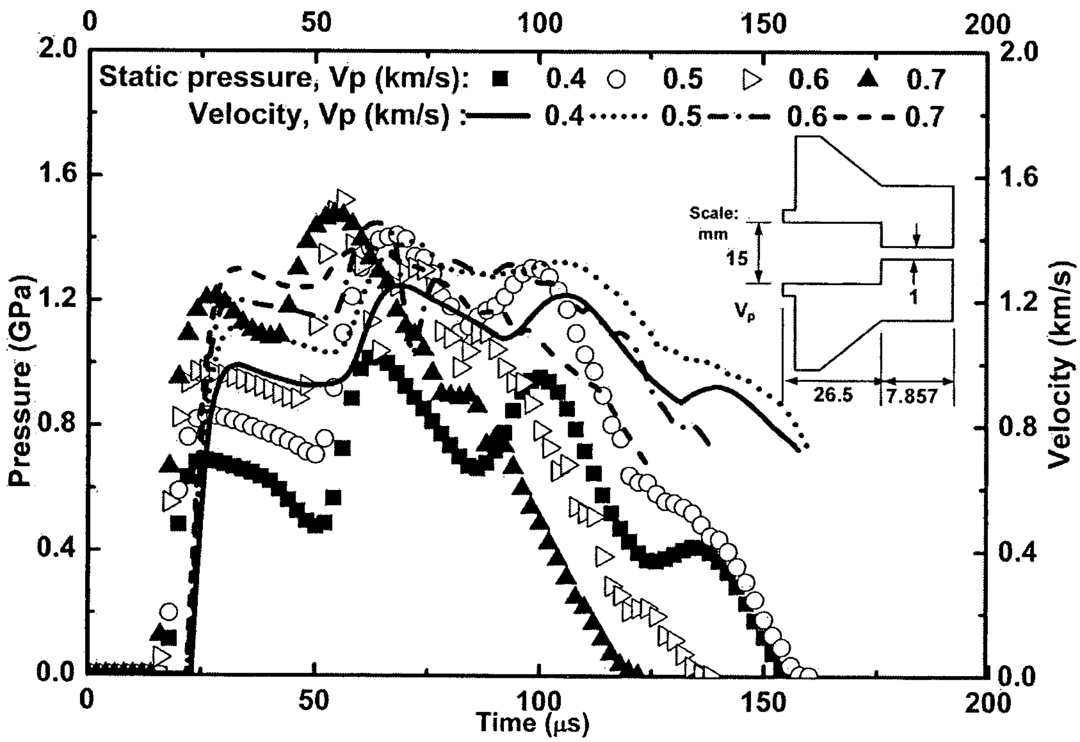


Fig. 10. Injection pressure and velocity with various projectile velocities

Table

Table 1
Fluid properties

Liquid	Bulk modulus (GPa)	Vapor pressure ($\times 10^{-6}$ GPa)	Molecular weight (g)	Specific heat (J/(kg.K)	Surface tension coefficient (N/m)
Water	2.24	3.17	18	4,182	0.0717
Diesel	1.6	1.378	170	1,850	0.0244
Kerosene	1.43	0.513	167	2,090	0.0235

Table 2
Validation with experiments of Mattujak et al. [7][a]

Results from:	Peak pressure (GPa)			Duration (μs)	
	1st	2nd	3rd	$\Delta t_{1\text{-}2}$	$\Delta t_{2\text{-}3}$
Exp	1.24	0.60	0.27	45.08	38.82
CFD	1.10	0.40	0.30	20	57

[a]Conical nozzle containing the water and 0.3 km/s of projectile velocity

Table 3
Validation with works of Pianthong et al. [10, 12]

Results from:	Maximum jet velocity (km/s)			
	Conical nozzle[a]			Stepped nozzle[b]
	Water	Diesel	Kerosene	Diesel
Exp	1.34	1.36	1.32	1.24
CFD	1.49	1.42	1.39	1.25

[a] 0.3 km/s of projectile velocity
[b]0.7 km/s of projectile velocity

Table 4
Validation with works of Shi [16][a]

Results from:	Maximum pressure (GPa) with brass piston mass:			Jet velocity (km/s) with brass piston mass:		
	0.38 g	0.58 g	0.75 g	0.38 g	0.58 g	0.75 g
Exp	0.14	0.11	0.10	0.50	0.48	0.38
CFD	0.16	0.15	0.13	0.52	0.50	0.40

[a]Generating the diesel jet by using MEM with 0.41 km/s of projectile velocity