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. Two 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 quite 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 works of 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 shown 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.

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.

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
\[
\int_{t_0}^{t_1} dV = \int_{t_0}^{t_1} \left( \frac{F(t)}{m} \right) dt
\]  

(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

\[
V(t) = V(t-\Delta t) + (F(t)/m) \Delta t
\]  

(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 \( \rho \) 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>
<thead>
<tr>
<th>Liquid</th>
<th>Bulk modulus (GPa)</th>
<th>Vapor pressure (Pa)</th>
<th>Surface tension (N/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>2.24</td>
<td>3,169</td>
<td>0.0717</td>
</tr>
<tr>
<td>Diesel</td>
<td>1.6</td>
<td>1,378</td>
<td>0.0244</td>
</tr>
</tbody>
</table>

\[
\rho_i = \frac{\rho_v}{1.0-(P_i-P_v)/B}
\]  

(3)

\[
a_i = \frac{1-(P_i-P_v)}{B} \times \frac{\sqrt{B}}{\rho_v}
\]  

(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 size (dt) of 0.1 microseconds was set; therefore, results from each calculation can be recorded. Turbulence model is the standard k-ε 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.

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.
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 (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


