20th Australasian Fluid Mechanics Conference
Perth, Australia
5-8 December 2016

Free-surface Flow Simulation of Unlike-doublet Impinging Jet Atomization

J. Kouwa[1], S. Matsuno[2], C. Inoue[3], T. Himeno[3] and T. Watanabe[3]

[1]Heat & Fluid Dynamics Department, IHI Corporation,
1, Shin-Nakahara-Cho, Isogo-ku, Yokohama 235-8501, Japan

[2]Research Department, IHI INC.,
150 East 52nd Street, 24th Floor, New York, NY 10022, U.S.A.

[3]Department of Aeronautics and Astronautics, The University of Tokyo,
7-3-1, Hongo, Bunkyo-ku, Tokyo 113-8656, Japan

Abstract

In liquid-propellant chemical propulsion systems, the liquid fuel and oxidizer are atomized by impinging jet atomization, mixed and combustions will occur due to auto-ignition inside a chamber. It is important for a performance prediction to simulate the primary atomization phenomenon; especially, the local mixture ratio can be used as indicator of thrust performance, so it is useful to evaluate it from numerical simulations. In this research, to predict local mixture ratio distribution downstream from an impingement point, we propose a numerical method for considering bi-liquid and the mixture and install it to CIP-LSM which is a two-phase flow simulation solver with level-set and MARS method as an interfacial tracking method. A new parameter, β, which is defined as the volume fraction of one liquid in the mixed liquid within a cell is introduced and the solver calculates the advection of β, inflow and outflow flux of β to a cell. SMART method is used for the interpolating value in a cell. By validating this solver, we conducted a simple experiment and the same simulation. From the result, the solver can predict the penetrating length of a liquid jet correctly and it is confirmed that the solver can simulate the mixing of liquids. Then we apply this solver to the numerical simulation of impinging jet atomization. From the result, the inclination angle of fan after the impingement in the bi-liquid condition reasonably agrees with the theoretical value. Also, it is seen that the mixture of liquids can be simulated in this result. We validate the numerical method by comparing numerical results with the experimental results with local mass flux and mixture ratio distributions.

Introduction

Liquid-Propellant chemical propulsion systems are often used for thrusters of space satellites to control attitude or orbit. Especially bi-propellant thrusters which utilize hypergolic ignition of fuel and oxidizer are preferred for higher performance and reliability. Figure 1 shows the conceptual diagram of a bi-propellant thruster. The liquids are atomized and mixed by impinging jet atomization. So far, designing the injector of a thruster tends to rely on the designer's experiences and numerous experiments conducted in the past. However, the amount of experiments using actual liquid propellants such as MMH or NTO does not suffice to investigate the large variety of thinkable injector design options. Also some kinds of propellants are difficult to handle due to their toxicity. Therefore, a design method using numerical simulation has been desired. The spray characteristics downstream such as distributions of mass flux, local mixture ratio, droplet diameter and velocity will vary with jet conditions or even farther upstream condition. So, it is important for an accurate evaluation to simulate the primary atomization phenomenon; especially, the local mixture ratio can be used as indicator of thrust performance as shown in Ref. [1]), so it is useful to evaluate it from numerical simulations.

So far, several researches about numerical simulations of impinging jet atomization have been conducted. For example, Zheng et al. conducted a numerical simulation of a like-doublet impinging rocket injector with CLSVOF method to investigate the primary atomization and evaluate the Sauter Mean Diameter D32 downstream. However, they focused on the detailed atomization process of liquid sheets and dealt with mono-liquid. Li et al. constructed a method of the Coupled Level Set and Volume of Fluid (CLSVOF) combined with Adaptive Mesh Refinement and calculated the droplet size distribution for impinging jet atomization[2]). Arienti et al., another member of the same research group as Li, applied CLSVOF to a liquid-propellant rocket engine; they considered two liquids, but they are immscible and they didn't evaluate local mixture ratio. However, actual bi-propellant thrusters use two kinds of liquid as mentioned above and it is important for injector designs to evaluate local mixture ratio downstream under non-reactive conditions[3]).

Therefore, we conduct numerical simulations of impinging jet atomization by using CIP-LSM and its modified codes to evaluate mass flux distribution and local mixture ratio distribution down. Then, we validate the solver by a simple experiment. Finally, we apply the solver to an impinging jet atomization to investigate the mixing of liquids downstream and understand the mechanism of mixing.

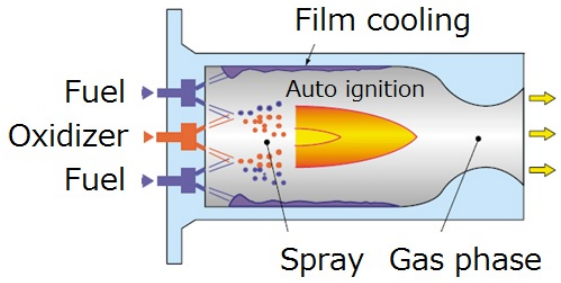


Figure 1: Conceptual diagram of bi-propellant thrusters

Methodology

Numerical Simulation

CIP-LSM(CIP based Level-Set & MARS) solver as proposed by Himeno et al.[4, 5, 6]) is used to simulate the atomization phenomena of impinging jet nozzles. In this solver, TCUP(Temperature-based CIP-CUP) method[7]) calculates the governing equations of fluid. The algorithm is shown in Fig. 2. The level-set method[8]) captures the gas-liquid phase interface. MARS(Multi-interface Advection and Reconstruction

Solver[9]) denotes the direct analysis method of multiphase flow with considering the gradient of the interface. Surface tension is calculated by the CSF method[10]). CIP-LSM is the coupling solver of these schemes. The governing equations are the Navier-Stokes equations for homogeneous two-phase flow as follows:

$$\begin{cases} \dfrac{D\rho}{Dt} = -\rho\nabla\cdot\vec{u} \\ \rho\dfrac{D\vec{u}}{Dt} = -\nabla p + \nabla(\mathbf{T}_v + \mathbf{T}_p) + \rho\vec{g} \\ \rho\dfrac{De}{Dt} = -p\nabla\cdot\vec{u} + \dot{\Theta} \end{cases} \tag{1}$$

where viscous stress tensor $\mathbf{T}_v$, interfacial surface stress tensor $\mathbf{T}_p$ and $\dot{\Theta}$ are defined as below:

$$\mathbf{T}_v = \lambda(\nabla\cdot\vec{u})\mathbf{I} + \mu(\nabla\vec{u} + \nabla\vec{u}^T) \tag{2}$$

$$\mathbf{T}_p = \sigma\delta_s(\mathbf{I} - \vec{n}_s\vec{n}_s) \tag{3}$$

$$\dot{\Theta} = (\mathbf{T}_v + \mathbf{T}_\sigma)\nabla\cdot\vec{u} - \nabla\cdot\vec{q} + \dot{L} \tag{4}$$

Here δ_s indicates the delta function which is defined only at the gas-liquid interface, $\vec{n}_s$ is a normal unit vector pointing to the liquid phase, $\mathbf{I}$ is the unit matrix.
To specify the fluid phase at a certain point in the flowfield, the level-set function ϕ is introduced, whose value gives the signed distance from the gas-liquid interface. Furthermore, the Heaviside function H_s is defined as follows:

$$\begin{cases} H_s = 0.5 & if \quad \phi > 0 \quad : liquid\ phase \\ H_s = 0 & if \quad \phi = 0 \quad : interface \\ H_s = -0.5 & if \quad \phi < 0 \quad : gas\ phase \end{cases} \tag{5}$$

Figure 3 shows he conceptual diagram of the level-set function and heaviside function.
In this research, phase change is not considered and each fluid particle remains in its initial phase attribution. Therefore the level-set function is advected according to the equation below:

$$\frac{\partial H_s}{\partial t} + (\vec{u}\cdot\nabla)H_s = 0 \tag{6}$$

Also, the fully laminar flow is assumed because the two jets are injected into a static fluid and hence effects of gas phase flow on gas-liquid interfaces is assumed to be negligible.

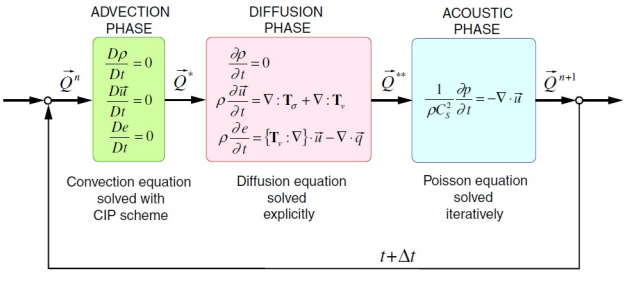


Figure 2: Numerical Algorithm of TCUP Method[4, 5, 6]

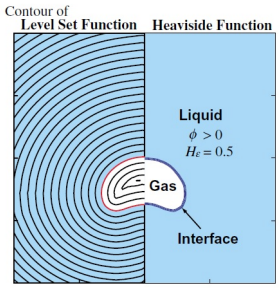


Figure 3: Conceptual diagram of level-set function and heaviside function[4, 5, 6]

Consideration of Mixture and Properties of Liquids

CIP-LSM is a two phase flow solver for mono-liquid. So, a modification is required to deal with two kinds of liquid properties and to consider mixing of each liquid. In the impinging jet atomization, assuming that the mixing due to advection is dominant, the mixing due to diffusion is ignorable. A parameter β is introduced which decides the volume fraction of one liquid in the liquid within a cell defined as follows.

$$\beta = \frac{V_{Liquid\#1}}{V_{cell}\times\alpha}, 1-\beta = \frac{V_{Liquid\#2}}{V_{cell}\times\alpha} \tag{7}$$

$$V_{Liquid\#1} + V_{Liquid\#2} = V_{cell}\times\alpha \tag{8}$$

Where, V_{cell} is the cell volume, β is the volume fraction of liquid in a cell and $V_{Liquid\#1}$ and $V_{Liquid\#2}$ are the volumes of Liquid #1 and #2 in the liquid phase respectively. To calculate the advection of β, an advection equation for β has to be solved. In addition to this, the inflow flux and outflow flux of β to a cell are computed.
In this method, the spatial gradient of a liquid in a cell which contains another liquid is not simulated, which means that inflow liquid to a cell is mixed completely at each iteration.

Simple Validation

To validate the conservation and physical applicability of the solver, a simple experiment and the same simulation are conducted. The experiment is that the coloring water jet by methylene blue dye is poured to a beaker that water is received as shown in Fig. 4. The high-speed camera, Photron SA-5, is used to visualize the motion of coloring water dynamically. The injecting speed of coloring water is 0.5m/s. The photographing frequency is 2,000Hz.

Figure 5 shows the visualizing and numerical result in t=0.055sec. after the injection. The evaluation is conducted by the penetration length of poured jet water because the value indicates the strength of advections. According to the results, the penetration lengths in the experimental and numerical result are 22.5mm and 22.3mm, respectively. Therefore, the solver can simulate the phenomenon of poured jet penetrations. However, The width in the experimental result is larger than the numerical one. This is because the numerical simulation cannot predict the mixing due to the turbulence of jets. In impinging jet atomizations, advections will be dominant for the mixing of liquids in impinging. So, we attempt to apply this solver to simulations of the impinging jet atomization under unlike-doublet conditions.

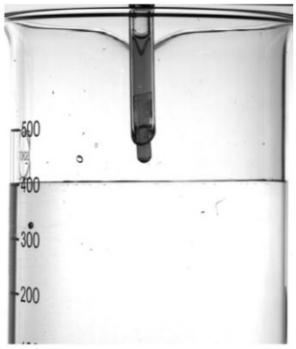
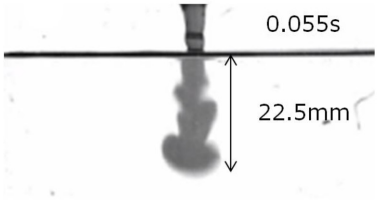
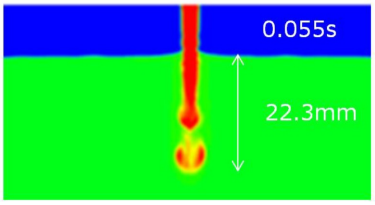


Figure 4: Experimental setup of simple validation



(a) Experimental result



(b) Numerical result

Figure 5: Results of the simple validation

Numerical Simulation on Impinging Jet Atomization

Figure 6 and 7 shows the numerical model and computational grid for numerical simulations of impinging jet atomization. The computational domain is 30mm × 36mm × 80mm and the number of grid is 111 × 101 × 131 points and the total is about 1.5 million grid points. Velocity-inlet condition is given to the jet portion and outflow condition is given to the other wall. The calculation starts from static fluid. The numerical condition of this numerical simulation is as shown in Tab. 1. Two cases are conducted, one is like-on-like condition and another is unlike-doublet condition.

Figure 8 shows the numerical results of fan angle after the impingement. According to the result, the fan angle of case1 is 90 degree because the jet momentum is the same as each other. On the other hand, the fan angle of case2 which is bi-liquid condition is 94.5 degree because the jet momentum of jet #1 is greater than jet #2 due to the difference of liquid density. The liquid sheet is leaned to the jet #2 side because the jet #1 pushes jet #2. To check the validity of the fan angle, the theoretical angle is calculated by using equation below:

$$\tan\beta = \frac{\dot{w}_1 v_1 \sin\theta_1 - \dot{w}_2 v_2 \sin\theta_2}{\dot{w}_1 v_1 \cos\theta_1 + \dot{w}_2 v_2 \cos\theta_2} \tag{9}$$

As the result, the fan angle from theoretical formulation is 94.7 degree and the numerical result almost agrees with the theoretical value. Figure 9 shows the time-series numerical results showing the gas-liquid interface colored by the mixture ratio. We can see that the mixture ratio changes with flow directions. No mixtures of liquid can be seen around the impingement. However, liquids will be mixed completely downstream from impingement. From this result, we can evaluate local mixture ratio distribution downstream numerically.

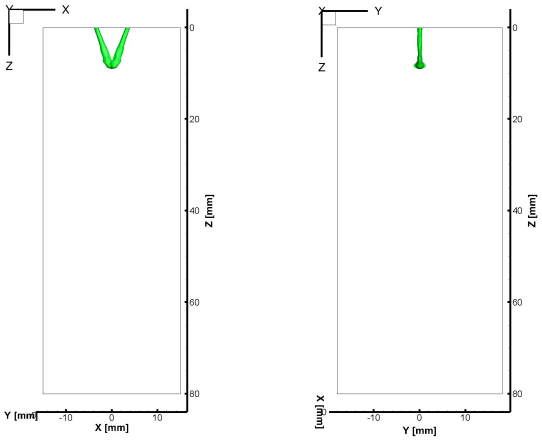


Figure 6: Numerical model

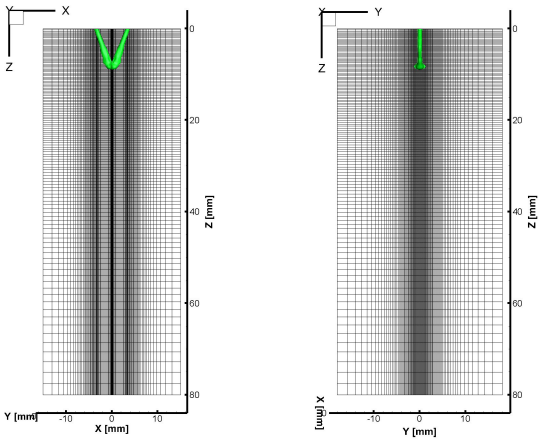


Figure 7: Computational grid

Table 1: Numerical condition

	Density		Diameter		Velocity		We number	
	A	B	A	B	A	B	A	B
	g/cm^3		mm		m/s		-	
case1	1.0	1.0	1.0	1.0	16.4	16.4	3697	3697
case2	1.4	1.0	1.0	1.0	16.4	16.4	5176	3697

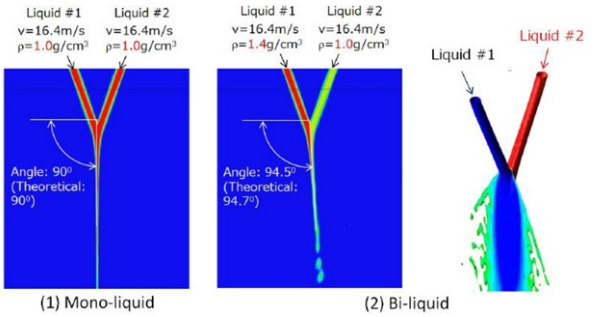


Figure 8: Numerical results of like-on-like and unlike-doublet condition

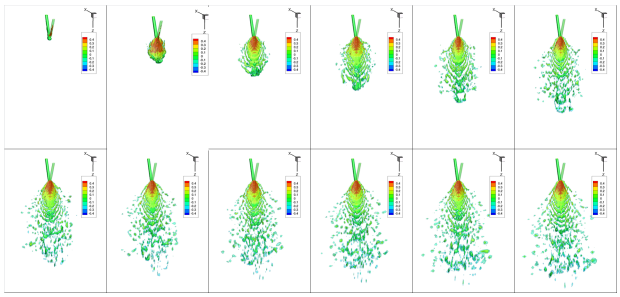


Figure 9: Time-series numerical results of mixture of liquids

Numerical simulation on a simulant thruster

In an actual thruster, some pairs of fuel and oxidizer jets are arranged on the injector circumferentially. To investigate the interference between the adjacent fans, we conduct the simulation of a simulant thruster. The diameter ratio R_D is 1.3 and the momentum ratio R_M is 1.2. The nine elements are arranged on a injector.

Figure 10 shows the result of a simulant thruster. From the result, we can see that the liquids scatter to radial direction due to the interference between fans. This phenomenon cannot be seen in simulations of single element. Furthermore, the contour plot at the downstream from impingement point indicates the mass flux distribution. From the result, a larger amount of liquid streamed around the interference point. So, the droplet diameter may increase. Therefore, we will investigate the change of droplet size distribution.

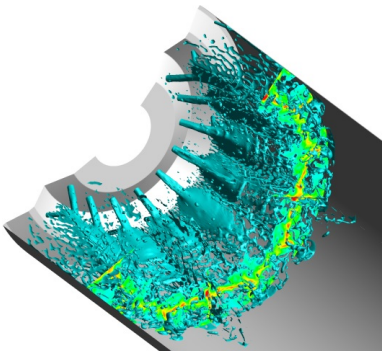


Figure 10: Simulation result of a simulant thruster

Conclusions

In order to predict local mixture ratio downstream to evaluate thruster ' s performance, we modify CIP-LSM developed by the University of Tokyo and validate the solver by conducting a simple experiment and simulation. As the result, the solver can predict mixture of liquid due to flow advection correctly. Therefore, we apply the solver to impinging jet atomization. From the result, we can see that the solver can be used to evaluate local mixture ratio distribution.

References

[1] J. H. Rupe, An Experimental Correlation of the Nonreactive Properties of Injection Schemes and Combustion Effects in a Liquid-Propellant Rocket Engine Part I. The Application of Nonreactive Spray Properties to Rocket Motor Injector Design, NASA Technical Report, No. 32-255, 1965

[2] X. Li, M. Arienti, M. C. Soteriou and M. Sussman, Towards an Efficient, High-Fidelity Methodology for Liquid Jet Atomization Computations, AIAA paper 2010-0210, 2010

[3] M. Arienti, X. Li, M. C. Soteriou, C. A. Eckett, M. Sussman, R. Jensen, Coupled Level-Set/Volume-of-Fluid Method for the Simulation of Liquid Atomization in Propulsion Device Injectors, 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference Exhibit

[4] T. Himeno, T.Watanabe, S. Nonaka, Y. Naruo and Y. Inatani, Numerical Analysis of Sloshing andWave Breaking in a Small Vessel by CIP-LSM, *JSME Int. J., Series B*, Vol. 47, No. 4, 2004, pp.709-715

[5] T. Himeno and T. Watanabe, Thermo-Fluid Management Under Low-Gravity Conditions: 2nd Report, Freesurface Flows Driven by Surface Forces, *Transaction of the Japan Society of Mechanical Engineers, Series B*, Vol. 68, No. 687, 2003, pp. 2400-2407

[6] T. Himeno, T. Watanabe and A. Konno, Numerical Analysis of Two-Phase Flow Behavier in a Liquid Propellant Tank, AIAA Paper 99-2177, 1999

[7] T. Yabe and P. Y. Wang, Unified Numerical Procedure for Compressible and Incompressible Fluid, *Journal of the Physical Society of Japan*, Vol. 60, No. 7, 1991, pp.2105-2108

[8] M. Sussman, P. Smereka and S. Osher, A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow, *Journal of Computational Physics*, Vol. 114, 1994, pp. 146-159

[9] S. Osher and J. A. Sethian, Front Propagating with Curvature-Depending Speed: Algorithms Based on Hamilton-Jacobi Formulations, *Journal of Computational Physics*, Vol. 79, 1988, pp.12-49

[10] J. U. Brackbill, D. B. Kothe and C. Zemach, A Continuum Method for Modeling Surface Tension, *Journal of Computational Physics*, Vol. 100, 1992, pp. 335-354