Paper ID ICLASS06-182

TWO-DIMENSIONAL SIMULATION ON ATOMIZATION IN A ROCKET ENGINE COAXIAL INJECTOR FLOW

Kazuya SHIMIZU[1] and Taro SHIMIZU[1]

[1,2]JAXA's Engineering Digital Innovation Center, Japan Aerospace Exploration Agency
3-1-1, Yoshinodai, Sagamihara, Kanagawa, 229-8510 Japan
[1]shimizu.kazuya@jaxa.jp, [2]shimizu.taro@jaxa.jp

ABSTRACT Numerical simulation code for rocket engine injector flow has been developed in order to investigate atomization and mixing of oxygen with hydrogen, and clarify the combusting flow in a liquid rocket engine combustor. In the simulation code, equation of state for the real fluid can be strictly considered and incompressible and compressible flow can be solved at the same time. Simulation results on the vibration of a single liquid drop agree well with theoretical ones. The simulation results also successfully show the breakup of liquid jet with inlet fluctuation. It is confirmed from these facts that numerical method used in this study is correct and can be applied to simulation on atomization and mixing in a liquid rocket engine injector.

Keywords: CFD, two phase flow, liquid rocket engine, injector, atomization

1. INTORDUCTION

It is necessary to understand the combusting flow in the high pressure condition in order to improve the efficiency and reliability of liquid rocket engine. One of the features of rocket engine combusting flow is the high pressure condition which reaches about 200atm, it is difficult to clarify the combustion flow experimentally. In that condition, to make flow field clear with numerical simulation is expected and computational fluid dynamics will take a very important role. Additionally, in order to understand flow filed in a rocket engine combustor, it is necessary to establish unified simulation method for atomization of liquid oxygen jet, mixing of oxygen with hydrogen and combusting flow under the high pressure condition. This is because, for example, that combustion oscillations in the rocket engine may be caused by unsteady phenomena of atomization or mixing near the injector. Furthermore the improvement of a coaxial injector depends on experiences or experimental trial and errors while the design principle of a coaxial injector is found in Ref. [1] or Ref. [2]. These facts result from less comprehension or understanding of coaxial injector flow with gas-liquid two phase flow and combustion.

Recently, the fast progress in terms of CFD technique for gas-liquid two phase flow has made it possible to simulate interaction between liquid and gas with surface tension[3][4][5][6][7]. In many cases of the simulation mentioned above, the numerical method is based on the scheme for incompressible flow, so there is the assumption of fixed density of liquid or low Mach number in a target of analysis. However, there are three requirements to simulate rocket engine coaxial injector flow. First, incompressible and compressible flow must be calculated at the same time. In the rocket engine injector, hydrogen velocity reaches about 400m/s and is several tens times as high as liquid oxygen velocity. The atomization and mixing strongly depend on the velocity ratio. It is important to clarify the effect of shear flow instability on the atomization. Second, the equation of state for real fluid must be strictly considered in the high pressure condition, because the assumption of ideal gas or the fixed density of liquid oxygen is not appropriate under the high combusting pressure of about 200atm. As the pressure and temperature approaches the critical point, it is necessary to include strict equation of state. Finally, numerical technique for two phase flow in which the surface tension can be taken into account is required, since there exists surface tension at the interface between oxygen and hydrogen owing to lower temperature than the critical temperature of oxygen. In this study, therefore, the particular attention is paid on calculation of the atomization and mixing, as intending to establish unified simulation method for atomization of liquid oxygen jet, mixing of oxygen with hydrogen and combusting flow under the high pressure condition in the future. The numerical simulation code which satisfies the three requirements mentioned above (calculating incompressible and compressible flow at the same time, including equation of state for real fluid and considering surface tension) has been developed in the two-dimensional cylindrical coordinate system and the validity of numerical method has been confirmed with basic gas-liquid two phase flow.

2. NUMERICAL METHOD

The governing equations used in this work consist of hydrodynamic equations (1)-(3) with non-conservative form including surface tension term, and the advection equation (4) of color function ϕ which is used in order to chase the phase boundary between gas and liquid and to estimate surface tension. In addition, Eq. (5) is used as equation of state. Eq. (3) on temperature is derived from the conservation law of energy, equation of state (5) and thermodynamic relations. These equations can be applied to any kinds of real fluid, because there is no assumption like ideal gas in the process to lead these equations[8][9].

$$\frac{\partial}{\partial t}\rho + \vec{u}\cdot\nabla\rho = -\rho\nabla\cdot\vec{u} \tag{1}$$

$$\frac{\partial}{\partial t}\vec{u} + \vec{u}\cdot\nabla\vec{u} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} + \frac{1}{\rho}\vec{F}_{st} \tag{2}$$

$$\frac{\partial}{\partial t}T + \vec{u}\cdot\nabla T = -\frac{T}{\rho c_v}\left(\frac{\partial p}{\partial T}\right)_\rho \nabla\cdot\vec{u} + \frac{1}{\rho c_v}\boldsymbol{\Phi} + \frac{1}{\rho c_v}\nabla\cdot\vec{q} \tag{3}$$

$$\frac{\partial}{\partial t}\phi + \vec{u}\cdot\nabla\phi = 0 \tag{4}$$

$$\Delta p = \left(\frac{\partial p}{\partial \rho}\right)_T \Delta\rho + \left(\frac{\partial p}{\partial T}\right)_\rho \Delta T \tag{5}$$

The governing equations are split into the three stages of advection phase, diffusion phase and acoustic phase by a time splitting technique. Advection equations consisting of Eqs. (6)-(9) are solved at the first stage by the CIP method[10][11][12][13] which has proven less diffusive and stable.

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

$$\frac{\partial}{\partial t}\vec{u} + \vec{u}\cdot\nabla\vec{u} = 0 \tag{7}$$

$$\frac{\partial}{\partial t}T + \vec{u}\cdot\nabla T = 0 \tag{8}$$

$$\frac{\partial}{\partial t}\phi + \vec{u}\cdot\nabla\phi = 0 \tag{9}$$

In the CIP method spatial profile within a grid cell between the two neighboring grid points is interpolated with a cubic polynomial, and both the values (density, velocity, temperature and color function) and its spatial derivatives are predicted in advance consistently with the master equation. The method has an advantage for the multi phase flow in which the steep gradient of density exists between the phase boundary.

At the second stage, Eqs. (10)-(13) which includes the diffusion term and thermal conduction term are solved. The turbulence viscosity and turbulence thermal conductivity are estimated in this stage, too.

$$\frac{\partial}{\partial t}\rho = 0 \tag{10}$$

$$\frac{\partial}{\partial t}\vec{u} = \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} \tag{11}$$

$$\frac{\partial}{\partial t}T = \frac{1}{\rho c_v}\boldsymbol{\Phi} + \frac{1}{\rho c_v}\nabla\cdot\vec{q} \tag{12}$$

$$\frac{\partial}{\partial t}\phi = 0 \tag{13}$$

We can easily adopt the implicit scheme in the diffusion phase because the equations solved in the diffusion phase are linear owing to extracting only diffusion term. First order implicit method is used in this work.

Equations (14)-(17) which have the term related to the pressure are solved by the CUP method[14] in the acoustic phase at the last stage.

$$\frac{\partial}{\partial t}\rho = -\rho\nabla\cdot\vec{u} \tag{14}$$

$$\frac{\partial}{\partial t}\vec{u} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\vec{F}_{st} \tag{15}$$

$$\frac{\partial}{\partial t}T = -\frac{T}{\rho c_v}\left(\frac{\partial p}{\partial T}\right)_\rho \nabla\cdot\vec{u} \tag{16}$$

$$\frac{\partial}{\partial t}\phi = 0 \tag{17}$$

In the CUP method, the pressure is implicitly predicted by the Poisson's equation (18). After that, density, velocity and temperature are updated by using the pressure predicted from the Poisson's equation.

$$\nabla\cdot\left(\frac{1}{\rho}\nabla p^{n+1}\right) = \frac{1}{\Delta t^2}\frac{p^{n+1}-p}{\rho C_s^2} + \frac{\nabla\cdot\vec{u}}{\Delta t} + \nabla\cdot\left(\frac{1}{\rho}\vec{F}_{st}\right) \tag{18}$$

$$C_S^2 \equiv \left(\frac{\partial p}{\partial \rho}\right)_T + \frac{T}{\rho^2 c_v}\left(\frac{\partial p}{\partial T}\right)_\rho^2 \tag{19}$$

When the sound speed approaches to the infinity, Eq. (18) changes to the Poisson's equation (20) which is usually used in an analysis of incompressible flows like MAC method[15].

$$\nabla\cdot\left(\frac{1}{\rho}\nabla p^{n+1}\right) = \frac{\nabla\cdot\vec{u}}{\Delta t} + \nabla\cdot\left(\frac{1}{\rho}\vec{F}_{st}\right) \tag{20}$$

This fact means the CUP method can treat both compressible and incompressible flow at the same time. The surface tension is predicted from CSF model[16], and estimated in the acoustic phase because the surface tension needs to match with the pressure gradient between the phase boundary.

3. RESULTS AND DISCUSSIONS

3.1 A single droplet oscillation

Firstly, calculation on a liquid drop vibration was carried out, in order to confirm the numerical method and estimation of surface tension. Numerical results are shown in Fig. 1. Liquid is water and gas is air in this test. IAPWS95[17] is used for equation of state of water and air is treated as an ideal gas. A non-spherical liquid droplet will become spherical after oscillating some times by the surface tension. The time transient is calculated in this

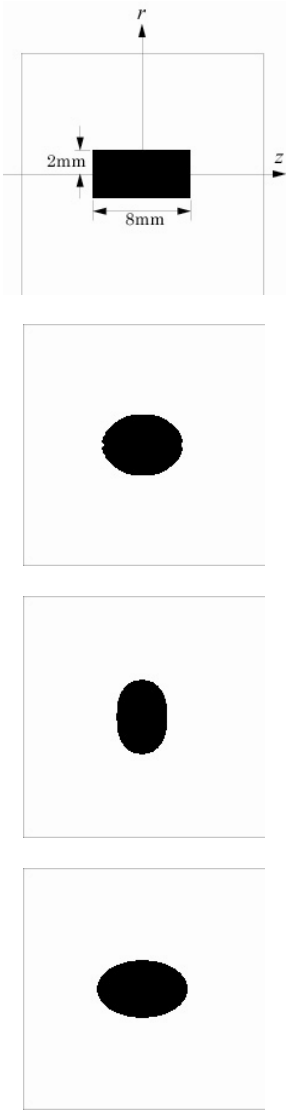

Fig. 1: Single droplet vibration simulation. The plotted time is 0msec, 100msec, 200msec and 300msec from top to bottom (white: air, black: water).

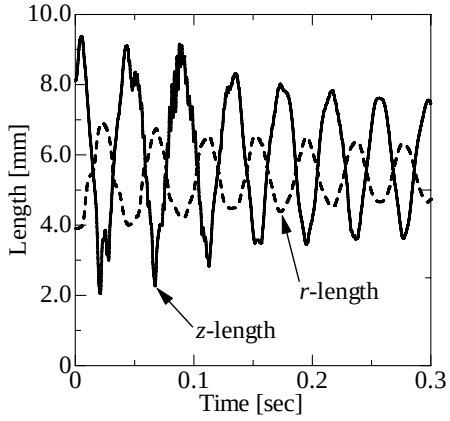

Fig. 2: The time evolution of the length of a single water drop.

numerical test. The initial distribution is rectangle in the cylindrical coordinate system shown in Fig. 1. This means cylinder droplet is arranged for the initial condition as considered three-dimensionally. Figure 2 shows the time evolution of the droplet length on r and z axes. The oscillation is reduced by viscosity dissipation, but in this case the many times of oscillations are needed to make droplet spherical because the viscosity is very small. The period (τ) of a scintilla oscillation of a single droplet is predicted from linear theory. The period is expressed as Eq. (21) when the radius of droplet is r[18].

$$\tau = \pi \sqrt{\frac{\rho_l r^3}{2\sigma}} \tag{21}$$

Substituting the radius, which gives the same volume as the cylinder drop shown in the top of Fig. 1, gives the period of 40.6msec. From Fig. 2, the period estimated numerically agrees well with the period predicted theoretically. This fact suggests that estimation of the surface tension and convection should be correct not only qualitatively but also quantitatively.

3.2 Liquid jet instability

Next, the simulation on the breakup of liquid jet flowing into the stationary gas has been carried out. Figure 4 shows one of the typical calculation results about the instability and the breakup of a liquid jet into the stationary gas. The calculation region and conditions are shown in Fig. 3. The inlet velocity (u_{in}) is fluctuated by Eq. (22) under the conditions that u_0 and A are set for 1.0m/s and 0.05 respectively.

$$u_{in} = u_0 + u_0 A \sin\left(\frac{2\pi t}{\tau}\right) \tag{22}$$

τ is the period of the fluctuation, and in this study the breakup length is examined for various fluctuation periods. Velocity fluctuation and surface tension make the instability and cause the breakup of a liquid jet. The dependency of the breakup length on the wave length (λ) of the fluctuation of the inlet velocity is shown in Fig. 5. The wave length is defined as the expression of Eq. (23).

$$\lambda = u_0 \tau \tag{23}$$

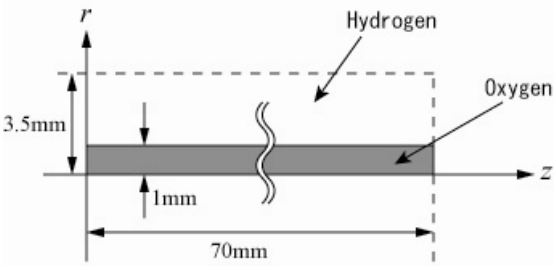

Fig. 3: Calculation region.

In this study, the shortest breakup length of 28.0mm is obtained for the fluctuation period of 5.0msec. Theoretical analysis from the Weber's theory predicts the shortest breakup length of the 21.0mm for the fluctuation period of the 8.9msec. Since a little disagreement is observed as comparing calculation results with the theoretical ones, further investigation may be needed.

Fig. 4: Instability and breakup of liquid jet (white: gas, black: liquid). The plotted time is 0msec, 20msec, 40msec, 60msec, 80msec and 100msec from top to bottom.

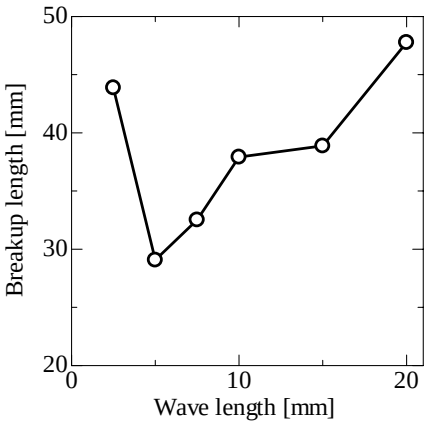

Fig. 5: Breakup length against wave length of the inlet fluctuation.

The simulation on the instability and breakup of liquid jet also has been carried out under the condition that inlet gas velocity is the same as and higher than the inlet velocity of liquid jet. It is shown in Fig. 6 that a liquid droplet after breakup is more spherical than a droplet shown in Fig. 4. It is observed that each interval of droplets is almost the same. When inlet gas velocity is higher than that of liquid jet, a

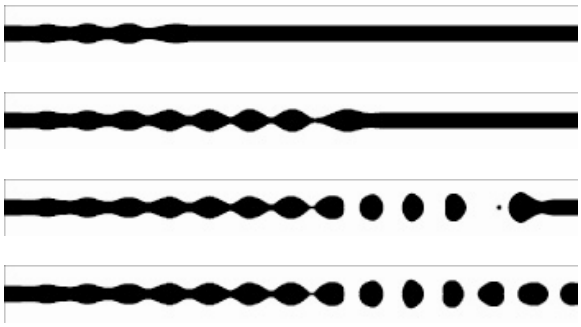

Fig. 6: Breakup of liquid jet. The inlet velocity of gas is the same as the inlet velocity of liquid jet. The plotted time is 20msec, 40msec, 60msec and 80msec from top to bottom.

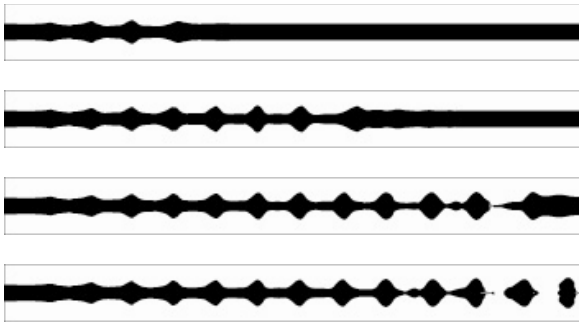

Fig. 7: Breakup of liquid jet. The inlet velocity of gas is higher than that of liquid jet. The plotted time is 20msec, 40msec, 60msec and 80msec from top to bottom.

droplet after breakup becomes non-spherical and the breakup length is longer than that of low inlet velocity of gas phase, as shown in Fig. 7.

4. CONCLUSION

Numerical simulation code has been developed in order to clarify flow field with atomization of liquid oxygen, mixing of oxygen with hydrogen, and combustion in a liquid rocket engine combustor. In the simulation code, the surface tension can be estimated, equation of state for all kinds of real fluid can be considered strictly and incompressible and compressible flow can be calculated at the same time. The validity of numerical method has been investigated by a basic gas-liquid two phase flow. It has been proven that the surface tension and convection should be predicted correctly from a numerical test of a single liquid droplet oscillation. And numerical tests of the instability and breakup of liquid jet suggest that this code can be applied to the unified simulation for atomization, mixing and combusting flow. The calculation for the injector flow with high velocity ratio of hydrogen to oxygen and turbulent model are going to be carried out in the future.

NOMENCLATURE

c_v : Specific heat at constant volume [J/kg K]

$\vec{F}_{st}$: Surface tension vector [N]

p : Pressure [Pa]

$\vec{q}$: Thermal conduction vector [W/m^2]

T : Temperature [K]

$\vec{u}$: Velocity vector [m/s]

Φ : Energy dissipation function

ϕ : Color function

τ : Viscosity stress tensor

ρ : Density [kg/m^3]

σ : Surface tension coefficient [N/m]

REFERENCES

[1] R. J. Burick, NASA CR-120936, "Space Storable Propellant Performance Program Coaxial Injector Characterization" (1972)

[2] NASA SP--8089, "Liquid Rocket Engine Injector" (1976)

[3] C. W. Hirt et al, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries", *Journal of Computational Physics*, Vol.39, pp.201—225 (1981)

[4] S. Osher et al, "Fronts Propagating with Curvature-Dependent Speed Algorithm based on Hamilton-Jacobi Formulation", *Journal of Computational Physics*, Vol.79, pp.12—49 (1988)

[5] Takashi Yabe et al, "Description of Complex an Sharp Interface during Shock Wave Interaction with Liquid Drop", *Journal of Physical Society of Japan*, Vol.62, No.8, pp.2537—2540 (1993)

[6] S. Koshizuka et al, "Moving-Particle Semi-Implict Method for Fragmentation of Incompressible Fluid", *Nuclear Science and Engineering*, Vol.123, pp.421—434 (1996)

[7] T. Kunugi et al, "SLUG-PLUG Flow Analyses of Stratified Flows in a Horizontal Duct by Means of the MARS", *Proceedings of the 5th ASME/JSME Joint Thermal Engineering Conference*, AJTE99-6435 (1999)

[8] Feng Xiao et al, "An Efficient Model for Driven Flow and Application to a Gas Circuit Breaker", *Computer Modeling and Simulation in Engineering*, Vol.2, pp.235—249 (1996)

[9] Takashi Yabe et al, "The Constrained Interpolation Profile Method for Multiphase Analysis", *Journal of Computational Physics*, Vol.169, pp.556—593 (2001)

[10] H. Takewaki et al, "Cubic Interpolated Pseudo-particle Method (CIP) for Solving Hyperbolic-type Equations", *Journal of Computational Physics*, Vol.61, pp.261—268 (1985)

[11] T. Yabe et al, "A Universal Solver for Hyperbolic Equations by Cubic-polynomial Interpolation, I. One--dimensional Solver", *Computer Physics Communications*, Vol.66, pp.219—232 (1991)

[12] T. Yabe et al, "A Universal Solver for Hyperbolic Equations by Cubic-polynomial Interpolation, II. Two-- and Three--dimensional Solvers", *Computer Physics Communications*, Vol.66, pp.233—242 (1991)

[13] F. Xiao et al, "Constructing a Multi-dimensional Oscillation Preventing Scheme for the Advection Equation by a Rational Function", *Computer Physics Communications*, Vol.94, pp.103—118 (1996)

[14] Takashi Yabe et al, "Unified Numerical Procedure for Compressible and Incompressible Fluid", *Journal of the Physical Society of Japan*, Vol.60, No.7, pp.2105—2108 (1991)

[15] Francis H. Harlow et al: "Numerical Calculation of Time-dependent Viscous Incompressible Flow of Fluid with Free Surface", *The Physics of Fluids*, Vol.8, No.12, pp.2182—2189 (1965)

[16] J. U. Brackbill et al, "A Continuum Method for Modeling Surface Tension", *Journal of Computational Physics*, Vol.100, pp.335—354 (1992)

[17] W. Wagner et al, "The IAPWS Formulation 1995 for the Thermodynamic Properties of Ordinary Water Substance for General and Scientific Use", *Journal of Physical and Chemical Reference Data*, Vol.31, No.2, pp.387—535 (2002)

[18] Basil N. Antar et al, "Fundamentals of Low Gravity Fluid Dynamics and Heat Transfer", CRC Press (1993)