JASMAC Conference of the Japan Society of Microgravity Application

P24

将来型宇宙機向けサーモダイナミクスベントシステムに関する研究 (ミキシングジェット挙動の数値解析)

A Study on Thermodynamic Venting System for Future Space Vehicles (Numerical Analysis of Mixing Jet Behavior)

坪内聡汰[1], 今井良二[1], 河南治[2],
Sota Tsubochi[1], Ryoji IMAI[1], Osamu KAWANAMI[2]
1 室蘭工業大学，Graduate School of Muroran Institute of Technology,
2 兵庫県立大学，Graduate School of University of Hyogo

1. Introduction

Currently, space exploration organizations such as ISECG are planning to use the lunar surface as a gateway for further exploration into deep space as a future plan[1)]. This plan is expected to increase the length of missions and the payload of the transportation system compared to conventional space development. For this reason, it is planned to install a propulsion system with high specific impulse that uses liquid hydrogen (LH2) or liquefied natural gas (LNG) as fuel and liquid oxygen (LOX) as oxidizer[1)]. However, when these propellants, which are classified as cryogenic liquids, are stored for long missions, the heat input from sunlight or the engine causes the propellants to evaporate as their temperature rises, resulting in an increase in tank pressure. Accordingly, spacecraft operated in long-distance space transportation systems require propellant tank pressure control.

Therefore, the final objective of this study is to develop a TVS (Thermodynamic Vent System) as a method to realize long-term storage of cryogenic propellant by managing and controlling the thermal hydraulic behavior inside and outside the propellant tank[2)]. So far, ground-based experiments have been conducted to demonstrate the effectiveness of jet mixing, one of the elemental components of the TVS[2)].

However, as this research progresses, it is necessary to conduct microgravity experiments assuming operation in the space environment. It is difficult to conduct such experiments more than once due to the large amount of money and labor involved. If it is possible to establish a model that accurately predicts jet mixing behavior using CFD, it would streamline the development of TVS by jet mixing. Therefore, the goal of this study was to establish a predictive model of jet mixing on the basis of CFD.

2. Theory and Methodology

Fig. 1 shows a schematic diagram of TVS. On a cruising spacecraft, acceleration is applied (propellant retention) in the direction of the bottom of the propellant tank as viewed from the propellant tank, in order to retain liquid propellant in the propellant outlet. If the gas-liquid interface reaches saturation temperature, a large amount of BOG is generated and the tank pressure rises. In jet mixing, liquid propellant taken from a tank is given a subcooling degree by a refrigerator and injected through a nozzle at the bottom of the tank to cool the gas-liquid interface by forced convection to suppress BOG.

Jet mixing development and verification can be made more efficient by creating a model that accurately predicts jet behavior on numerical analysis and incorporating it into development.

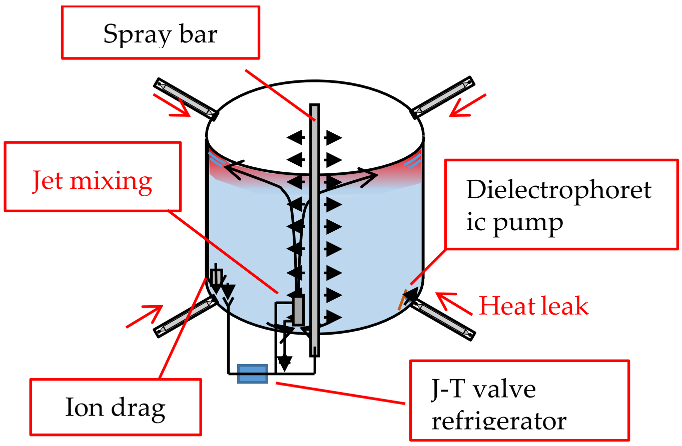


Figure.1 TVS Overview

In order for evaluate and improve the jet behavior prediction model, this study compares the experimental values obtained by jet mixing experiments with those obtained by numerical analysis. The temperature values and jet behavior are used for comparison. Experiments were selected from experiments conducted by the University of Hyogo to verify the effectiveness of jet mixing[3]. The jets were selected because they were visualized in this experiment and the numerical results could be compared with the jet behavior.

3. Analysis Methods

Based on the aforementioned objectives, a 3D analytical model identical to the tank used in the experiment was created, and the values obtained in the experiment were input and analyzed. The basic equations are the equation of motion, the energy equation, the volume fraction (VOF) equation, and the continuity equation. The turbulence model used was k-ε Realizable. The analysis range is 0 seconds, which is the time when the jet started feeding into the tank in which the thermal stratification was formed in the liquid phase, and extends to the time when the jet feeding is terminated.

In addition, two experimental conditions were selected for this study. The numbering indicating experimental conditions conforms to that established by the University of Hyogo[3].

The Temperature Difference in **Table 1** is the temperature difference between the liquid surface temperature and the jet temperature.

Table.1 Experimental Condition

No.	1	2
Liquid Level [mm]	60	
Jet Supply Time [sec]	2	
Jet Flow Rate [mL/min]	102.58	70.16
Tempureture Difference [K]	30	
Test Fluid	water	

Next, the analytical conditions are described. The main analysis conditions are shown in **Table 2**. The analysis was performed on an inner tank, and a multiphase flow analysis including the gas phase was performed. Note that the temperature of the gas phase was set the same as the liquid surface temperature. In addition, the wall surface is in adiabatic condition, and heat input from outside is not considered. As for the physical properties of the fluid, the physical properties of air defined by ANSYS FLUENT are given as constant values for the gas phase, and for the liquid phase, density, viscosity, and specific heat are given as polynomials in terms of temperature, since the test fluid in the experiment is water.

Regarding the boundary conditions, all of them are velocity inlet and pressure outlet. In addition, two conditions were used for the initial values of turbulent kinetic energy k and turbulent specific extinction rate ε in the analysis domain. Case A uses the values obtained by the ANSYS FLUENT Compute function, and Case B sets both the turbulent kinetic energy k and the turbulent specific extinction rate ε to 0.

The Compute function is calculated using the transport equations for turbulent kinetic energy and turbulent dissipation rate from the flow velocity, pressure, temperature, and physical properties for each boundary condition.

Table.2 Analysis Condition

No.	1	2
Acceleration [mm]	y= -9.80665	
Velocity Inlet [m/s]	4.353630	2.977682
Pressure Outlet [Pa]	101325	
Tempureture Difference [K]	30	
Test Fluid	water	
Turbulent Kinetic Energy	Case A Case B	
Turbulent Specific Dissipation Rate		

4. Analysis Results

The aforementioned experimental and analytical conditions were entered and analyzed. The following **Fig.2, 3, and 4** compare the analytical and experimental results of No1, No2, respectively.

The graphs show the temperature change over time at each temperature sampling point. Each sampling point is distributed every 20 mm in height from the bottom of the tank. In addition, No1-A and No1-B in the graphs represent the analytical solutions obtained by applying the Case A and Case B conditions, respectively.

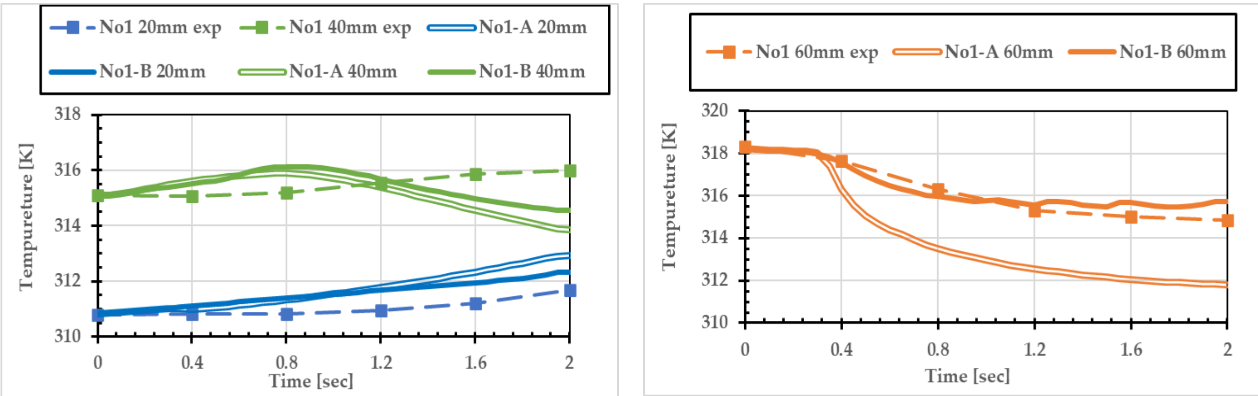


Figure.2 Comparison of experimental and analytical results (No1)

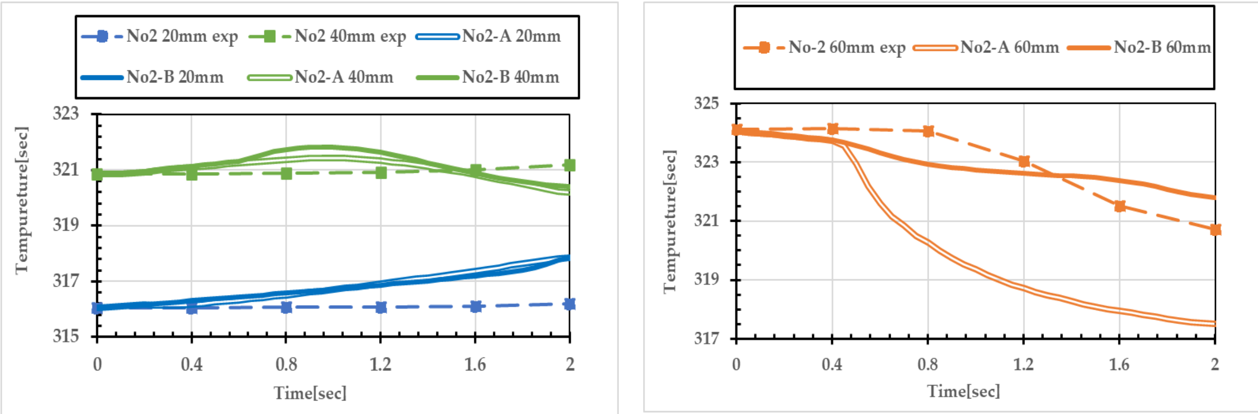


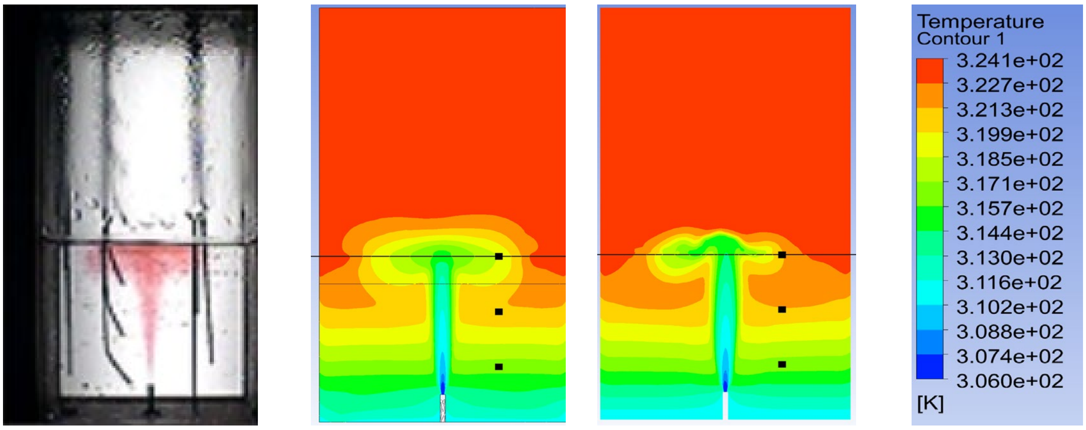
Figure.3 Comparison of experimental and analytical results (No2)

The graph shows that in No1 and No2, the analysis performed in Case B resulted in a smaller difference between the experimental and analytical values than the analysis performed in Case A. Especially at the liquid level, the difference between Case A and Case B is remarkable, with Case B obtaining analytical values that are extremely close to experimental values compared to Case A.

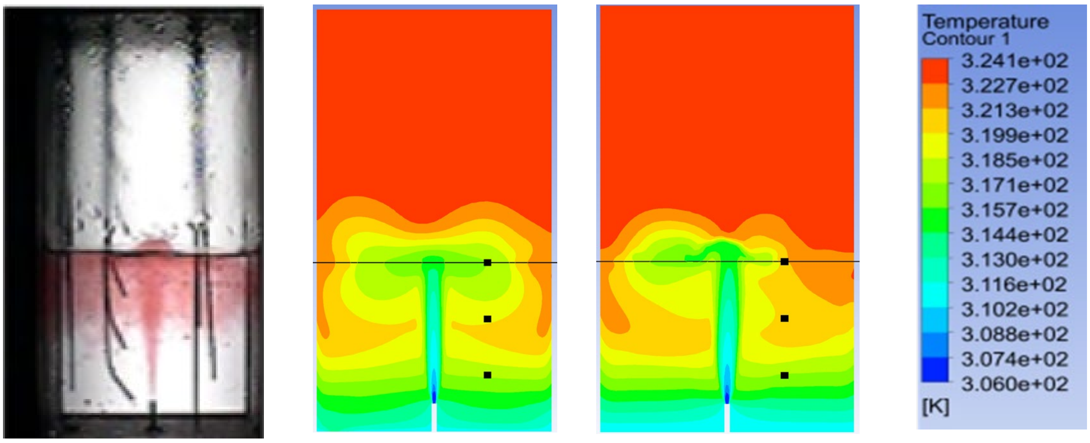
Next, **Fig.4** and **Fig.5** show a comparison of the images of jet behavior taken in the No1 and No2 experiments and the temperature contour plots obtained by the analysis. The black dots in the temperature contour diagram represent temperature measurement points, and the black lines represent liquid level height.

Note that the supercooled liquid supplied to the test tank in the experiment is colored red to visualize the jet behavior.

Focusing on the jet behavior in the experimental images, it can be seen that in No1 and No2, the jet reaches the liquid surface at 0.8sec and diffuses near the liquid surface at 1.6sec. Then, focusing on the temperature contour plots, in No1-A and B and No2-A and B, the jet reaches the liquid surface at 0.8sec and then widely cools near the liquid surface at 1.6sec, showing a similar behavior to the jet behavior in the experimental images. However, the temperature graphs for No1-A and No2-A show a large difference between the experimental and analytical values, indicating that a comparison of the experimental images and temperature contour plots cannot be used for a rigorous evaluation.



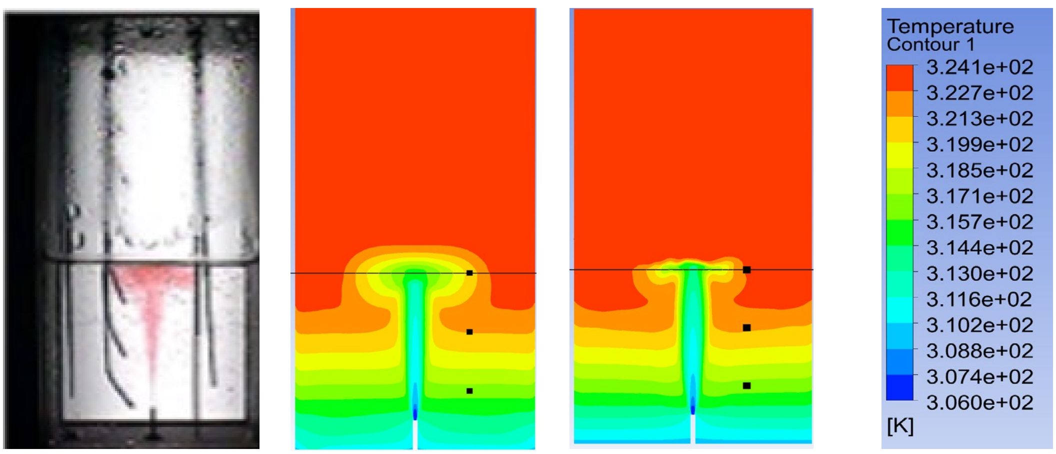
(a) 0.8sec



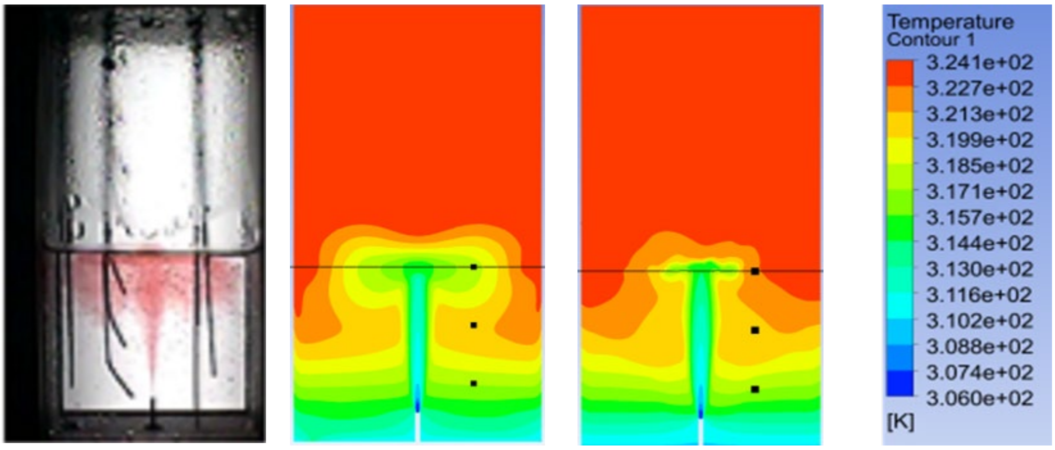
Experimentally Obtained Images of Jet Behavior | **No1-A** | **No1-B**

(b) 1.6sec

Figure.3 Comparison of No1 temperature contour and experimentally obtained images of jet behavior



(a) 0.8sec



Experimentally Obtained Images of Jet Behavior **No2-A** **No2-B**

(b) 1.6sec

Figure.4 Comparison of No2 temperature contour and experimentally obtained images of jet behavior

5. Discussion

The differences between Case A and Case B in the analysis results are discussed. As described in Analysis Methods, Case A and Case B have different initialization conditions. Case A was initialized with the default settings of ANSYS FLUENT, while Case B was initialized with the turbulent kinetic energy and turbulent dissipation rate set to 0.

Next, the liquid in the tank during the experiment is described. Since the experiment was conducted on the ground with the tank at rest, the liquid in the tank was close to a stationary fluid and the flow velocity was considered to be close to zero. Therefore, the turbulent kinetic energy and turbulent dissipation rate of the liquid in the tank are considered to be infinitesimally small. In the default initialization settings of ANSYS FLUENT, the turbulent kinetic energy k and the turbulent dissipation rate ε are calculated from the average over the entire analysis domain. Therefore, it is considered that the turbulent kinetic energy and turbulent dissipation rate were sometimes calculated excessively for the entire analysis domain as a result of considering the flow velocity defined at the velocity inlet.

As a result, we believe that turbulent viscosity increased throughout the analysis domain and heat transport during the analysis also increased. Therefore, in Case A, heat transport between the low-temperature fluid (jet) and the high-temperature fluid that formed a temperature stratification actively took place, and an excessive temperature drop near the liquid surface was considered to have occurred.

6. Conclusion

Based on the final objective of this study, which is to develop a TVS, a jet mixing prediction model was constructed. The method used to construct the predictive model was a comparison of experimental and analytical values obtained by jet-mixing experiments.

The results obtained in this study include the achievement of agreement between experimental and analytical values under specific analytical and experimental conditions, and the construction of a limited predictive model. However, the validity of the prediction model constructed in this study needs to be verified, since there were only a few cases in which a reasonable solution was obtained under specific analytical conditions.

References

1) The Global Exploration Roadmap 3rd Edition. (2018) , ISECG , Third Eition , pp1-29

2) Kazuaki.NISHIDA, Ryoji.IMAI, Osamu.KAWANAMI, Yutaka.UMEMURA, Takehiro.HIMENO: Ground-Based Experiment on Reducing Boil-Off Gas by Jet Mixing for Future Cryogenic Propulsion System. IJMSA ,Vol.38 , (2021) , Issue 1 , pp.1-6

3) Osamu KAWANAMI , Kentaro TAKEDA , Ryoki MATSUSHIMA , Ryoji IMAI , Yutaka UMEMURA3 and Takehiro HIMENO: Observation of Flow Pattern and Jet-tip Height Issuing from a Nozzle at Tank Bottom in Microgravity. IJMSA , Vol.39 , (2022) , Issue 1 , pp1-13