

Flow Characteristics in Rectangular and Circular Containers with Air Injected from a Slit Nozzle

T. KIWATA^{*1}, R. ASAI², K. ARIMA², Y. KAWAMURA³, H. UCHIDA³, and K. TOYODA⁴

1: School of Mechanical Engineering, Kanazawa University, Ishikawa, Japan

2: Graduate School of Natural Science and Technology, Kanazawa University, Ishikawa, Japan

3: ALTEMIRA Can Co., Ltd., Shizuoka, Japan

4: Hokkaido University of Science, Hokkaido, Japan

* Correspondent author: kiwata@se.kanazawa-u.ac.jp

Keywords: Container, Jet, Slit nozzle, PIV, Flow measurement, CFD

ABSTRACT

The jet flow injected into a container has been extensively studied by many researchers, as it is utilized for heating during the container manufacturing process. The present paper describes the flow characteristics of air injected from a slit nozzle into rectangular and circular containers. The effects of the length of the nozzle brims and the velocity of the planar jet on the flow in the containers were investigated numerically and experimentally. Velocity distributions were measured using the particle image velocimetry (PIV) system. Numerical simulations were performed by large eddy simulation (LES) using ANSYS Fluent 19.0. The length of the nozzle brims was varied from $b_n = 0$ mm to 52.5 mm. The Reynolds number of the jet flow, $Re (= U_0 b / \nu)$, where $b (= 5$ mm) is the nozzle width, U_0 is the nozzle exit velocity, and ν is the air kinematic viscosity, was varied from 2,000 to 6,000. The non-dimensional nozzle offset distance was set to $y_n/D = 0.3$, where $D (= 66$ mm) is the width (diameter) of the rectangular (circular) container. The contours of the time-averaged velocity by the numerical simulation are similar to those by the experiment. In the case of the rectangular container, $Re = 2,000$, the jet reaches near the bottom of the container when the length of the nozzle brims is $b_n > 20$ mm. For $Re = 6,000$, the jet reaches near the bottom of the container when the length of the nozzle brims is $b_n > 30$ mm. In contrast, in the case of the circular container, the injected planar jet into the circular container becomes three-dimensional, allowing the planar jet to reach the bottom of the circular container without the nozzle brims.

1. Introduction

Air jets are used in various applications, such as drying, heating, cooling, and dust removal (Hoof et al. 2012; Ben et al. 2017). In particular, the jet flow injected into a container has been extensively studied by many researchers, as it is utilized for heating during the container manufacturing process. Shakouchi et al. (1982) investigated experimentally and numerically the

flow of air injected from a slit nozzle inside a rectangular container. They clarified the oscillation of a two-dimensional jet injected from a nozzle located in the center of a cavity, and the relations between the occurrence of oscillation and the configuration of a cavity. Mataoui et al. (2001,2009) reported the flow characteristics of turbulent planar jets injected from nozzles inside a rectangular container by experiment and numerical analysis. They found that the oscillation of the jet in the container is affected by the irregular circulating vortices, which are changed with the Reynolds number. Noguchi et al. (2020) have investigated the effects of nozzle offset distance and Reynolds number on the behavior of planar jets injected into a rectangular container by experiments and numerical analysis. Although the planar nozzle, which is close to the container wall, does not allow the jet oscillation, the planar jet reaches the bottom of the rectangular container at high Reynolds numbers.

In this study, the results of the effects of the length of nozzle brims, which are installed at the nozzle exit, and the shapes of the container, which have a rectangular and a circular cross-section, on the flow characteristics in a container with air injected from a slit nozzle were experimentally and numerically investigated. The velocity distributions were measured using the particle image velocimetry (PIV) system. The numerical simulations were performed using ANSYS Fluent 19.0 based on the finite-volume method by the large-eddy simulation (LES).

2. Experimental apparatus and method

Figure 1 shows a schematic diagram of the cross-section of a container and a slit nozzle in the experimental setup. A centrifugal blower supplied the airflow to the settling chamber. The supplied air was rectified by passing through a perforated plate and flowing out into the test section as a planar jet from a slit nozzle with a span of $B = 330$ mm, a length of 32 mm, and a width of $b = 5$ mm. A tested container with square and circular cross-sections was placed below the slit nozzle. The rectangular container measured $D = 66$ mm in width and $H = 124$ mm in depth. The side wall of the rectangular container was made of transparent acrylic resin with a 1 mm-thick plate. The circular container also measured $D = 66$ mm in diameter and $H = 124$ mm in depth. The side wall of a circular container was made of a thin unreflective film. The origin of the coordinate axes was set at the center of the top of the container, and the container depth direction was defined as the positive direction of the x coordinate. The distance between the nozzle and the container mouth was $x_n = 25$ mm. The non-dimensional nozzle offset distance was fixed at $y_n/D = 0.3$. The Reynolds number of the jet was varied $Re (= U_0 b/\nu) = 2,000$ to 6,000, where ν is the air kinematic viscosity, and U_0 is the velocity at the center of the nozzle exit. The velocity distribution into the

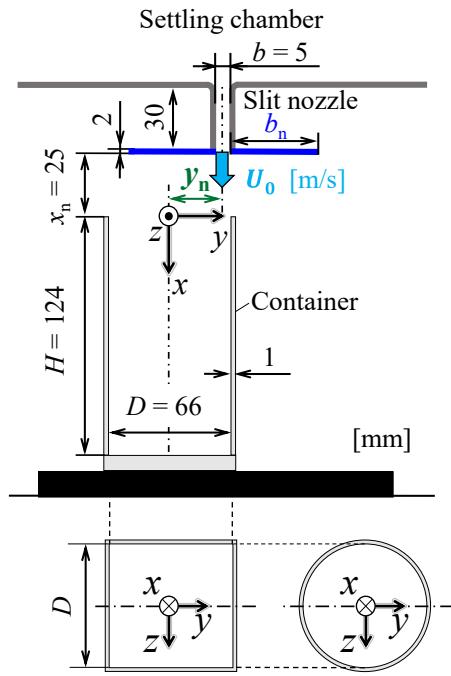


Figure. 1 Schematic diagrams of cross-section of a slit nozzle with a nozzle brim, rectangular and circular containers

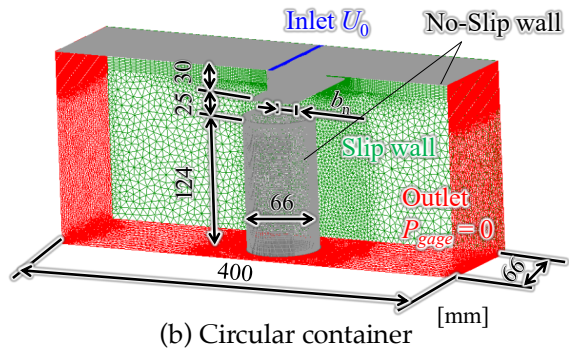
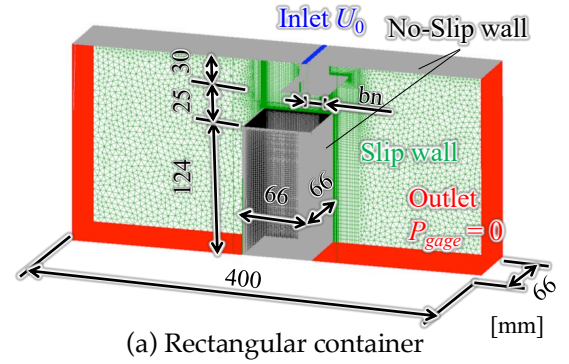


Figure. 2 Computational domain, mesh, and boundary conditions

container was measured using the particle image velocimetry (PIV) system. Tracer particles (Average diameter $\approx 1 \mu\text{m}$, Particle density $\approx 1,030 \text{ kg/m}^3$) were injected as an oil mist generated by a fog generator through a pipe diverted from the inlet of a centrifugal blower. A double-pulse Nd:YAG laser (Dantec Dynamics, Dual Power 200-15) was used to irradiate a laser sheet of approximately 1 mm thickness in the x - y plane. A total of 240 sets of 2 images for 30 seconds at 8 Hz using a CCD camera (Dantec Dynamics, Flow Sence EO) was taken for each case. The image size was $2,448 \times 2,050$ pixels, and the acquisition area was $287 \text{ mm} \times 240 \text{ mm}$. The instantaneous and time-averaged velocity vectors were calculated by the software of DANTEC Dynamics Studio Ver. 6.9.

3. Numerical setup

3.1 Governing equation and numerical method

The flow field was assumed to be viscous and incompressible. To provide comparisons with our experimental results, the numerical simulations were performed using ANSYS Fluent based on the finite-volume method with the LES model for the unsteady turbulent flow. The filtered Navier-Stokes and continuity equations were solved as the governing equations for LES. The

WALE model was used for the sub-grid scale model. The convection terms of the governing equation were discretized by the bounded central differencing scheme. Second-order central differencing was used for the diffusion term, while the second-order implicit scheme was used for the time marching. The pressure implicit with the splitting of the operator (PISO) method was used as the calculation algorithm for the pressure-velocity coupling.

3.2 Computational mesh and boundary conditions

Figure 2 shows the computational domain, mesh, and boundary conditions. To obtain accurate predictions, the mesh near the walls and nozzles was finely set. The size of the computational domain was set the same as the experiment, i.e., the streamwise, transverse, and spanwise lengths of the computational domain were 179 mm, 400 mm, and 66 mm, respectively. The computational mesh was created with a non-uniformly spaced structured mesh near the container and an unstructured mesh far from the container. The number of grid points for the rectangular and circular containers was about 2.6×10^6 and 3.2×10^6 , respectively. The boundary of the top and side of the computational domain satisfied the wall condition. A pressure outlet condition of $p = 0$ was specified at the downstream boundary of the domain.

4. Results and discussion

4.1 Effect of the length of nozzle brims

Figure 3 shows the contours of the time-averaged velocity $(\bar{u}^2 + \bar{v}^2)^{0.5}/U_0$ at the center cross-section of a rectangular container ($z = 0$) for the horizontal plate length of nozzle brims $b_h = 0$ mm, 20 mm, and 37.5 mm measured by the PIV. For $Re = 2,000$, when the slit nozzle has no brim ($b_h = 0$ mm), the planar jet does not reach the bottom of a container. This flow reverses its direction

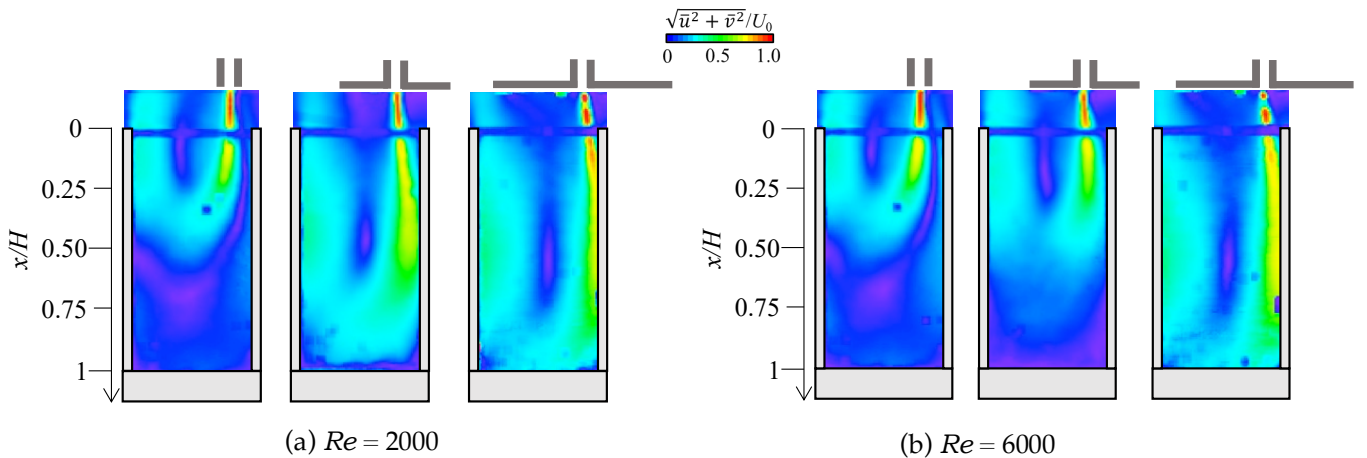


Figure. 3 Contours of time-averaged velocity by PIV (Rectangular container)

halfway down the container and exits from the left side of the top of the container. In contrast, for $b_n = 20$ mm and 37.5 mm, the jet flows along the right wall of the container due to the Coanda effect, turns near the bottom, travels upward along the left wall, and then exits from the left side of the top of the container. On the other hand, for $Re = 6,000$, when the slit nozzle has brims of $b_n = 20$ mm, the planar jet does not reach the bottom of a container. However, for $b_n = 37.5$ mm, the planar jet reaches the bottom of a container.

Figure 4 shows the distributions of velocity in the y -direction $\bar{v}/U_0$ at the center cross-section of the rectangular container for $Re = 2,000$. Note that the vertical positive axis is in the downward direction. The velocity profile has a negative peak. Therefore, to evaluate the reaching distance of the jet in the container, x_{max} was defined as the distance in the x -direction of the largest negative velocity in the y -direction, as shown in Figure 4. Figure 5 shows the variations of the reaching distance of the jet in the container, x_{max}/H , with the length of the nozzle brims, b_n . As the length of the nozzle brims, b_n , increases, x_{max}/H increases. For $Re = 2,000$, the jet reaches near the bottom of the container when the length of the nozzle brims is $b_n > 20$ mm. For $Re = 6,000$, the jet reaches near the bottom of the container when the length of the nozzle brims is $b_n > 30$ mm. The reaching distance of the jet in the container, x_{max}/H , is maximum for the length of the nozzle brims near $b_n = 37.5$ mm. From $b_n = 10$ mm to 30 mm, there is an influence of the Reynolds number. In particular, for $Re = 6,000$, the jet does not reach the bottom of the container unless the length of the nozzle brims is increased.

Figure 6 shows the contours of the time-averaged velocity $(\bar{u}^2 + \bar{v}^2)^{0.5}/U_0$ at the center cross-section of a rectangular container ($z = 0$) at $Re = 2,000$ by the CFD. The contours of the time-

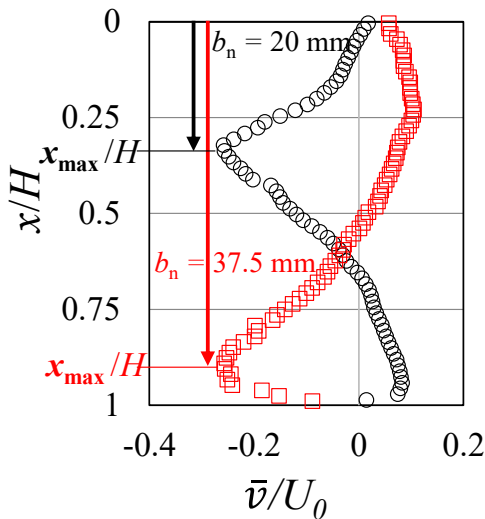


Figure. 4 Distributions of velocity in y -direction at the center cross-section of the rectangular container ($Re = 2,000$, Rectangular container)

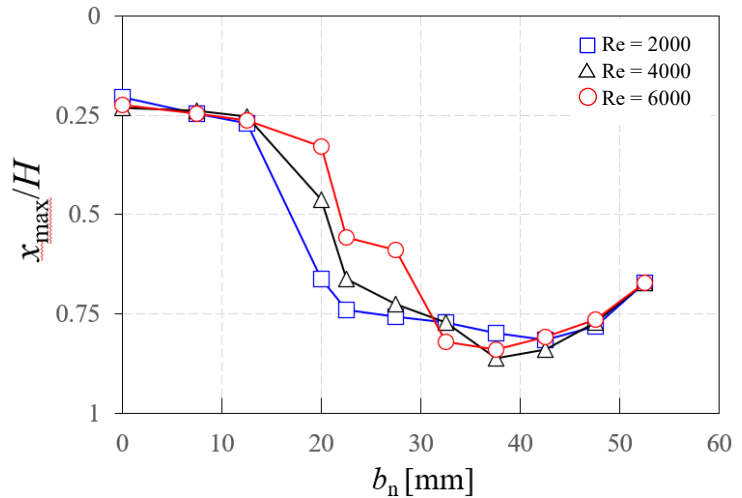


Figure. 5 Variation of x_{max} with the length of nozzle brims b_n (Rectangular container)

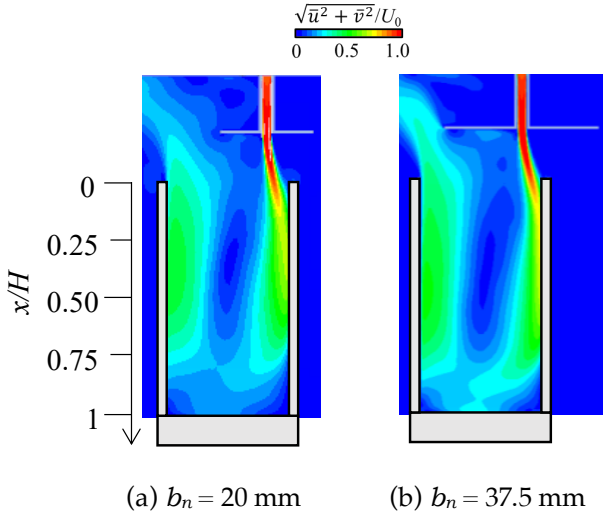


Figure. 6 Contours of time-averaged velocity by CFD ($Re = 2,000$, Rectangular container)

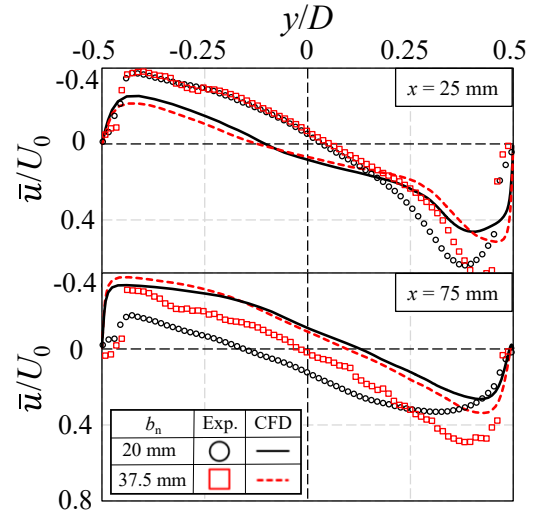


Figure. 7 Profiles of time-averaged x -axial velocity ($Re = 2,000$, Rectangular container)

averaged velocity by the numerical simulation are similar to those by the experiment of Figure 3(a). For $b_n = 20$ mm and 37.5 mm, the jet flow along the right wall of the container reaches the bottom of the container. Figure 7 shows the profiles of mean x -axial velocity $\bar{u}^2/U_0$ in the horizontal cross section at $x = 25$ mm ($x/H = 0.2$), and 50 mm ($x/H = 0.4$) for $Re = 2,000$. When the length of the nozzle brims becomes long, the peak velocity near the right-side wall is high, which is why the jet reaches the bottom of the container. The velocity distributions of the numerical simulation qualitatively agree with those of the experiment.

4.2 Effect of the shape of containers

Figure 8 shows the contours of the time-averaged velocity $(\bar{u}^2 + \bar{v}^2)^{0.5}/U_0$ at the center cross-section of a circular container ($z = 0$) with nozzle brims, and the nozzle offset distance was $y_n/D = 0.3$. The length of the nozzle brims was varied $b_n = 0$ to 37.5 mm. The results of the numerical simulation of $b_n = 20$ mm are also shown in Figure 8. In the case of no nozzle brim ($b_n = 0$ mm), the planar jet reaches the bottom of the container. It is different from the rectangular container as shown in Figure 3(a). There is no different from the flow pattern between $b_n = 20$ mm and 37.5 mm. For $b_n = 37.5$ mm, the contours of the time-averaged velocity by the numerical simulation are similar to those by the experiment.

Table 1 shows the contours of the time-averaged velocity $(\bar{u}^2 + \bar{v}^2 + \bar{w}^2)^{0.5}/U_0$ at the horizontal cross-section of rectangular and circular containers by the numerical simulation. The nozzle offset distance was $y_n/D = 0.3$. For a rectangular container, the shape of the injected planar jet maintains its linear shape near the right wall. Then, the jet flow descends along and parallel to the container wall. After reaching the bottom of the container, the jet flow ascends while remaining parallel to the wall, before exiting the container. On the other hand, for the circular container, the planar jet

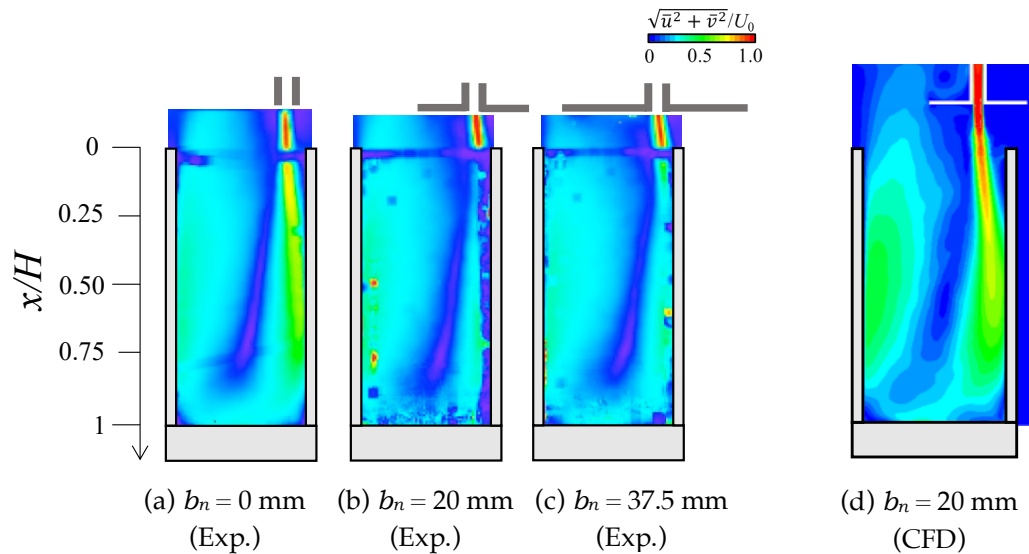


Figure. 8 Contours of time-averaged velocity by the experiment and the numerical simulation ($Re = 2000$, Circular container)

Table.1 Contours of time-averaged velocity at the horizontal cross-sections in rectangular and circular containers ($Re = 2000$)

$\sqrt{\bar{u}^2 + \bar{v}^2 + \bar{w}^2}/U_0$ 0 0.5 1	x/H	0	0.2	0.8
	Cylindrical Container			
	Rectangular Container			

maintains its planar shape at $x/H = 0$, the same as the rectangular container. However, at $x/H = 0.2$, it flows downward and spreads along the right concave wall. After reaching the bottom of the container, the jet changes direction and rises. The spread flow gathers at the other side of the wall, i.e., near the left wall of the circular container at $x/H = 0.8$. Consequently, the upward flows along the left wall converge toward the center of the circular container. This circulation makes the three-dimensional internal flow, allowing the jet to reach the container bottom even without nozzle brims.

5. Conclusion

The present study investigated the flow behaviour of a planar jet injected from a slit nozzle into rectangular and circular containers using the particle image velocimetry (PIV) system and the numerical simulation by the LES. It is found that the long nozzle brims are effective in attaching a slit jet to the wall due to the Coanda effect, and the planar jet reaches near the bottom of the rectangular container. On the other hand, the injected planar jet into the circular container becomes three-dimensional, allowing the planar jet to reach the bottom of the circular container without the nozzle brims.

Acknowledgments

The authors are thankful to Mr. Tomohiro Kuratani and the staff of Kanazawa University Technical Support Center for their help with the experimental apparatus manufacturing.

References

- Ben, K.R., Nejla, M.S., Hervé, B. & Georges, L.P., (2017). Effect of nozzle-to-plate spacing on the development of a plane jet impinging on a heated plate. *Journal of Heat Mass Transfer*, 53, 1305-1314.
- Hooff, T.V., Blocken, B., Defraeye, T., Carmeliet, J. & Heijst, G.J.F.V., (2012). PIV measurements and analysis of transitional flow in a reduced-scale model: Ventilation by a free plane jet with Coanda effect, *Building and Environment*, 56, 301-313.
- Mataoui, A., Schiestel, R. & Salem, A., (2001). Flow Regimes of Interaction of a Turbulent Plane Jet into a Rectangular Cavity: Experimental Approach and Numerical Modelling. *Flow, Turbulence and Combustion*, 67(4), 267-304.
- Mataoui, A. & Schiestel, R., (2009). Unsteady phenomena of an oscillating turbulent jet flow inside a cavity: Effect of aspect ratio. *Journal of Fluids and Structures*, 25, 60-79.
- Noguchi, K., Kiwata, T., Toyoda, K., Uchida, H., Tsuji, M. and Uegami, K., (2020). Flow characteristics in a rectangular container injected air from a slit nozzle. Proceedings of the 31st International Symposium on Transport Phenomena, Honolulu, USA, 1-7.

Shakouchi, T., Suematsu, Y. & Ito, T., (1982). A Study on Oscillatory Jet in a Cavity. *Bulletin of the JSME*, 25(206), 1258-1265.