Experimental and Computational Study on Drainage Flow in Rain Gutters of Buildings

Eiji Ejiri [1,*], Yutaro Oishi [2], Kiyokazu Iijima[3]

[1]Department of Mechanical Science and Engineering, Chiba Institute of Technology, Narashino, Japan
[2]Graduate School of Chiba Institute of Technology, Narashino, Japan
[3]Tanita Housingware Co. Ltd., Tokyo, Japan
*corresponding author: ejiri.eiji@it-chiba.ac.jp

Abstract The gas-liquid two-phase flow in the T-shaped channel of rain gutters of multistoried buildings was investigated both experimentally and computationally. A half-size miniaturized test section was used as an experimental model, in which the average flow changed direction from horizontal to vertical. The flow rate and pressure difference between immediately downstream of the rain gutter and the atmosphere were measured with a flowmeter and a monometer, respectively. A video of the internal flow was taken with a high-speed CCD video camera. Pressure, flow velocity, void fraction and other characteristics of the internal flow were computed by using a commercial CFD software code under boundary conditions corresponding to the experiments. The results showed that the computed total pressure loss was close to the experimental value and that the internal flow pattern in the vicinity of the branch was qualitatively almost the same as the one observed in the experiments. CFD analysis of the gas-liquid two-phase flow was shown to be useful in improving the drainage performance of the rain gutter channel. However, it is still very difficult to accurately predict the details of the two-phase flow. Further measurements of the two-phase flow are still needed to capture the detailed distribution of the void fraction in the internal flow.
Keywords: Rain gutter, Two-phase flow, Flow visualization, CFD

1 Introduction

Climate change due to global warming has tended to bring about bigger typhoons or local torrential downpours, resulting in unprecedented amounts of rainfall these days. There are stronger demands than ever to improve the drainage performance of rain gutters. An example is shown in Fig. 1 [1] of a rain gutter used on a building of multiple stories to gather rainwater falling on the roof and drain it downward. The internal flow in rain gutters often gives rise to gas-liquid two-phase flow as water mixes with air coming from upstream in the dead water region or the vortex [2]. Therefore, it is essential to understand the detailed characteristics of the internal channel flow in order to improve the drainage performance of rain gutters.

This study focused on the T-shaped channel section connecting the vertical downspout and the horizontal gutter, and the flow characteristics in the channel section were investigated experimentally and computationally. Experiments were carried out to measure the flow rate and pressure loss in the channel section and to visualize the gas-liquid two-phase flow with a high-speed CCD camera. The two-phase flow

Fig. 1 Example of a rain gutter used on buildings

was then analyzed numerically using a commercial thermo-fluid simulation code. The experimental and computational results were compared to clarify the flow mechanism of the channel section in order to obtain insights for improving the design of the channel section and replacing the conventional T-shaped channel.

2 Experiments

2.1 Experimental setup and method

The test section used in the experiments was a T-shaped circular pipe, which is shown in Fig. 2 in the form of a part drawing. It was just half the size of the actual product, and the inner diameter of the pipe was 40 mm. Flow came in from the right-hand side, made a right-angle turn and was discharged downward. A 10-mm-diameter hole was pierced in the top of the channel section, making it open to the atmosphere, which was originally intended to facilitate easy maintenance of the actual product. The hole permitted air to come in and mix with water, causing the gas-liquid two-phase flow. The test section was made of a transparent acrylic resin for visualization of the internal flow.

A system diagram of the experimental setup is shown in Fig. 3. Water was drawn up from a lower tank (Water tank 1) to an upper tank (Water tank 2) by a centrifugal pump. The water level in the upper tank was kept constant with a level sensor and an automatically controlled valve (Valve 1). Water flow was controlled with a downstream manual valve (Valve 2) and measured with an upstream electro-magnetic flowmeter. The pressure difference between the water surface in the upper tank and the exit of the test section was measured with a manometer. The total pressure loss coefficient of the test section, C_p, was evaluated with the following formula based on Bernoulli's equation:

$$C_p = \frac{\rho g h_m - (\rho v^2/2 + p_m)}{\rho v^2/2} \tag{1}$$

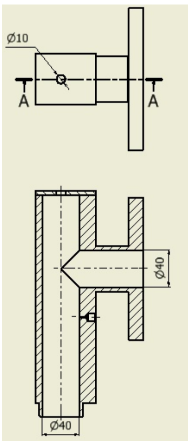


Fig. 2 T-shaped test channel section

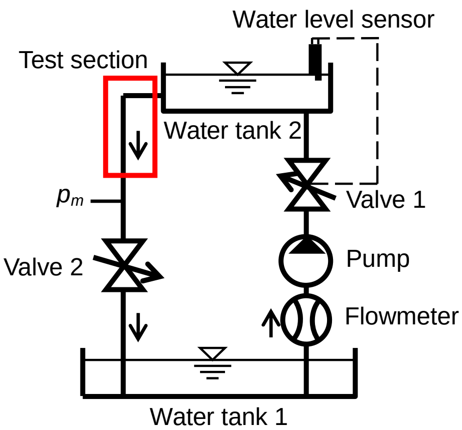


Fig. 3 System diagram of experimental setup

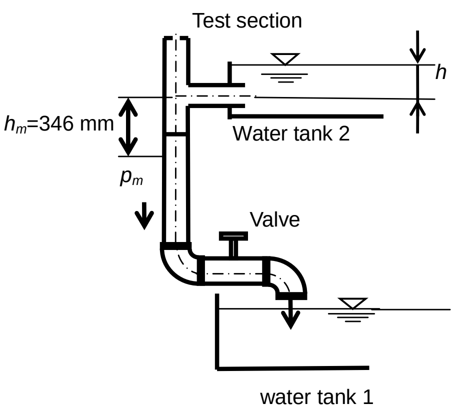
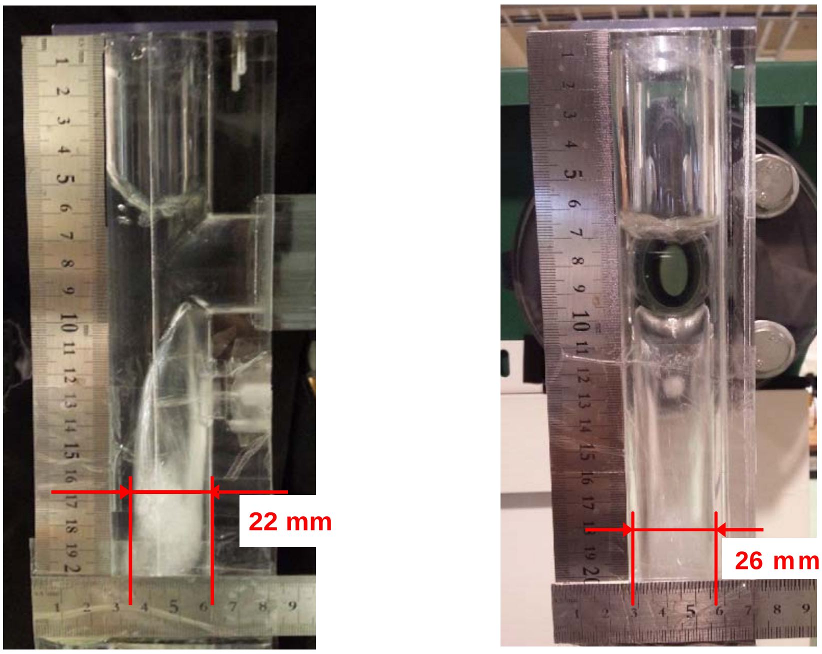


Fig. 4 Experimental conditions for water surface and pressure measurement

where, h_m, p_m, v, and ρ denote the height of the water surface in Tank 2 above the pressure measuring point, the measured pressure, average velocity in the pipe, and density of the fluid (water), respectively. Figure 4 shows the experimental conditions regarding the location of the pressure measuring point and the water levels.

2.2 Experimental results and discussion

The visualized flow in the test section is shown in Fig. 5. It is seen in Fig. 5(a) that the internal flow in the T-shaped channel section was a complex gas-liquid two-phase flow. Flow separated at the lower corner edge of the upstream pipe. A dead water region started from the edge and spread downstream. The width of the region was around 26 mm (Fig. 5(b)) and the maximum height from the wall was around 22 mm (Fig. 5(a)). The upper part of the region was hollow, however, the flow gradually became bubbly in the streamwise direction in the lower part. An intermittent vortex tube was also observed, connecting the dead water region and the upper water surface, as shown in Fig. 6. The average flow rate was 65 L/min and the total pressure loss coefficient, C_p, was 9.91 under a condition of h=80 mm.



(a) Side view (b) Front view

Fig. 5 Visualized flow in T-shaped model

Fig. 6 Visualized intermittent vortex tube in side view

3 Numerical analysis

3.1 Computational procedure

Unsteady three-dimensional two-phase flow computations were performed using a commercial thermo-fluid analysis code, SCRYU/Tetra V11, which employs the finite volume method. The analysis model was constructed based on the T-shaped test section used in the experiments. The total computational domain is shown in Fig. 7, and a part of the generated computational meshes is shown in Fig. 8. The total number of meshes was about 21×10^6. Meshes near the T-junction were made finer than those in other areas in order to obtain higher resolution in the computations there.

The boundary conditions in the computations were as follows. A flow rate of 65 L/min was set at the inlet and natural outflow was applied at the exit. Pressure was kept constantly at zero at the 10-mm-diameter hole in the top. In two-phase flow computations, it was assumed that the gas phase was incompressible air and that the liquid phase was incompressible water. The void fraction was assumed to be zero at both the inlet and exit of the computational domain. The Courant number was set at 1.0 for computational stability and the initial time interval was 1 ms.

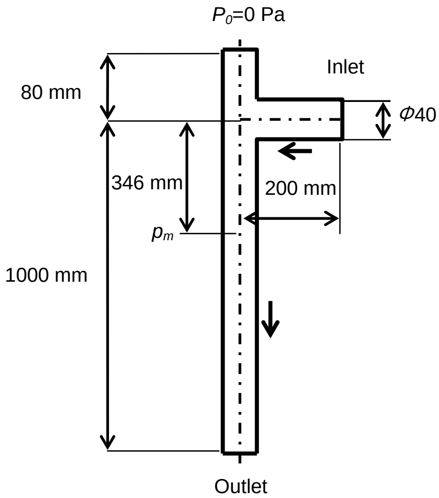


Fig. 7 Total computational domain

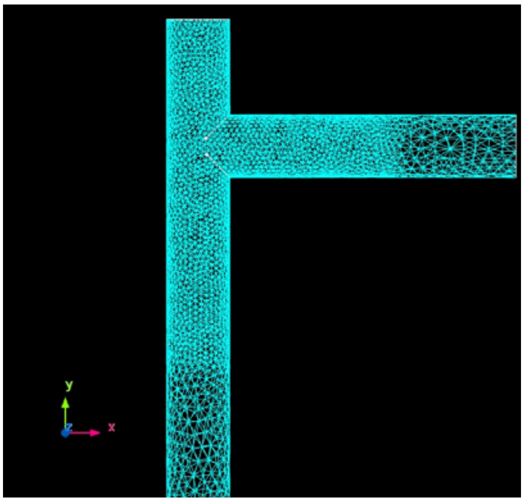


Fig. 8 Computational meshes

3.2 Numerical analysis results and discussion

Figure 9 shows the computed void fraction in a vertical cross section containing the center lines of the pipes. It is seen that the void fraction is 1 in the upper region of the vertical pipe and that the free surface of water is successfully simulated. The void fraction is close to 1 again at the downstream corner edge, which corresponds to the dead water region mentioned in subsection **2.2**. The width of the computed dead water region is around 23 mm, which is very close to the experimental result of 22 mm.

Figure 10 shows the computed void fraction in 10 horizontal cross sections, of which the distance, L, from the lower corner edge was increased by 10 mm from zero mm in the downstream direction. It is seen that the region of the high void fraction abruptly begins to grow from the lower corner edge and that the region is the largest at L=40 mm. The center of the region approaches the center of the pipe along the stream direction. Such a three-dimensional image of the dead water region cannot be easily obtained experimentally.

The free surface of water and the dead water region with the high void fraction were successfully simulated by the two-phase flow analysis, however, the intermittently generated vortex tube with rotation observed in the experiment could not be simulated in the analysis. It may be possible to simulate such a complicated flow phenomenon with a more sophisticated analysis technique such as the volume of fluid (VOF) method [3], which will require further study in the future.

The computed total pressure loss coefficient was 9.67, which was very close to the measured one, differing by only 3%. Therefore, the two-phase flow analysis used in this study is useful in designing new rain gutters and examining their flow patterns.

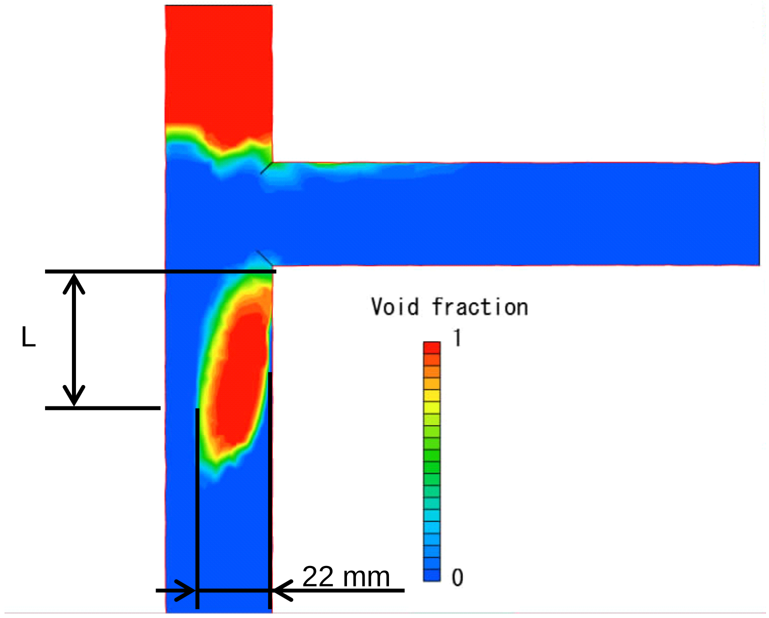


Fig. 9 Computed void fraction in cross section containing centerlines of pipes

5 Conclusions

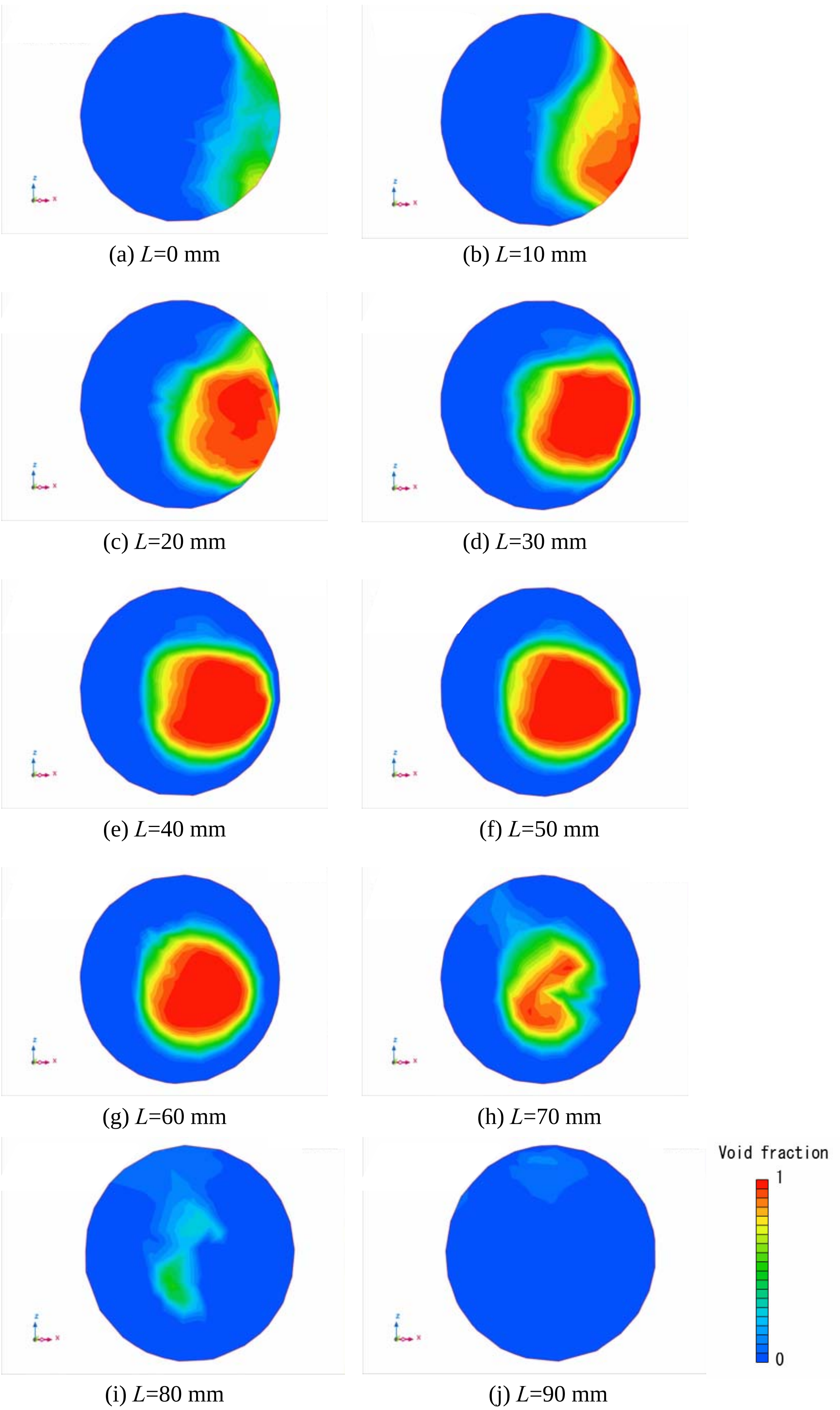
The following conclusions were obtained regarding the T-shaped flow channel of the rain gutters, based on the results obtained both by computation of a two-phase flow and by visualization and experimental measurements.
(1) The complex flow in the T-shaped test section was visualized with an experimental setup using a transparent acrylic resin. A free surface of water, a dead water region with bubbles and an intermittently generated vortex tube were clearly observed.
(2) The free surface of water and the dead water region were successfully simulated by two-phase flow computations using a commercial thermo-fluid software code, but the vortex tube was not.
(3) The computed size of the dead water region and the total pressure loss in the channel were very close to the measured values.
(4) The two-phase flow analysis method used in this study has sufficient accuracy for practical use. It can serve as a useful design tool useful for developing new flow channels.

References

[1] Tanita Housingware Co. Ltd. (2014) The construction collection of examples. Retrieved August, 2014, http://www.tanita-hw.co.jp/jirei/sj1-sid70.html.

[2] The Japan Society of Mechanical Engineers (2006) Handbook of gas-liquid two-phase flow technology. Coronasha, Tokyo (in Japanese).

[3] Yunliang C, Chao W, Bo W, Min D (2012) Three-dimensional Numerical Simulation of Vertical Vortex at Hydraulic Intake, Procedia Engineering, Vol. 28, pp 55-60.



(a) *L*=0 mm (b) *L*=10 mm

(c) *L*=20 mm (d) *L*=30 mm

(e) *L*=40 mm (f) *L*=50 mm

(g) *L*=60 mm (h) *L*=70 mm

(i) *L*=80 mm (j) *L*=90 mm

Fig. 10 Void fraction in cross sections for various values of *L*