BUBBLE BEHAVIOR IN LIQUID-GAS TWO-PHASE FLOW BEHIND A CROSS-SHAPED OBSTACLE IN A VERTICAL CIRCULAR DUCT

K. TAKASE*, G. KAWASAKI AND K. UETA
**Department of Nuclear System Safety Engineering, Nagaoka University of Technology, Japan*
E-mail: takase@vos.nagaokaut.ac.jp

Abstract: Grid spacers installed in subchannels of fuel assemblies for nuclear reactors can promote heat transfer. However, the fluid velocity and bubble behavior are greatly affected as the cross-sectional area of the flow passage changes. Therefore, the void fraction distribution behind the obstacle that simulates the grid spacer shape simply was measured by using a wire mesh sensor (WMS) system. Moreover, a two-phase flow analysis was performed to investigate the effect of the obstacle on the bubble behavior in a vertical duct.

Keywords: *Bubbler, Two-phase flow, Obstacle, Vertical duct, WMS, Experiment, Analysis.*

1. INTRODUCTION

Grid spacers [1] installed in subchannels of fuel assemblies for the BWR and PWR have the role of keeping the interval between adjacent fuel rods constantly. In addition, in case of PWR the grid spacer has also the role as the turbulence promoter. When the transient event occurs in PWR, two-phase flow is generated by boiling of water. Therefore, improving the reactor core thermal-hydraulic safety is important to confirm the bubbly flow behavior around the grid spacer. For the objective of clarifying the effect of an obstacle like the grid spacer to the bubbly flow, the void fraction behind the obstacle that simulates the grid spacer shape simply was measured by a WMS system and a two-phase flow analysis was performed to investigate the effect of the obstacle on bubble behavior.

2. EXPERIMENTAL STUDY

The experimental setup consists of a measuring section and water/air supply lines. The measuring section in Fig.1 is a vertical circular duct with a diameter of 58 mm made of an acrylic resin and an air injection nozzle with 120 injection holes with a diameter of 0.6 mm. A mixture of water and air flow from the bottom through the obstacle to the top of the duct.

The void fraction distribution at the horizontal cross-section in the duct is measured with a WMS system [2]. Figure 2 shows the WMS which consists of nine wires with a diameter of 0.3 mm. In the experiments, superficial velocities of water and air, J_L and J_G, were set to be bubbly flow or slug flow. An obstacle in Fig.3 with the dimensions of thickness 5 mm, height 50 mm and horizontal length 58 mm is installed in each position of 20, 100 or 180 mm upstream from the WMS.

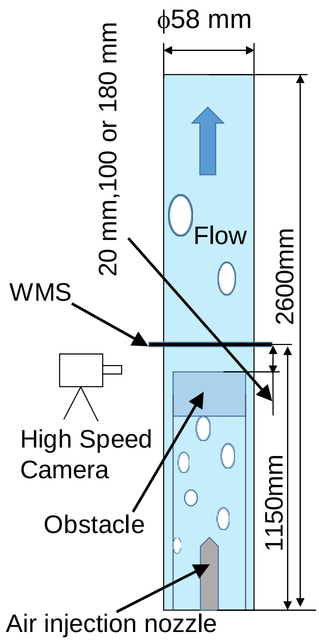


Fig.1 Experimantal duct

At the slug flow condition ($J_L=0.5$ m/s and $J_G=0.4$ m/s), the void fraction becomes low at the position just behind the obstacle and high at the other region (Fig.4(a)). This is because large bubbles are divided by the obstacle. In Fig.4 (b), (c) the void fraction becomes high at the core region because the divided bubbles coalesce with increasing the axial distance.


Fig.2 WMS

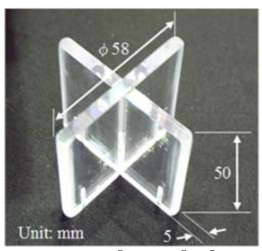
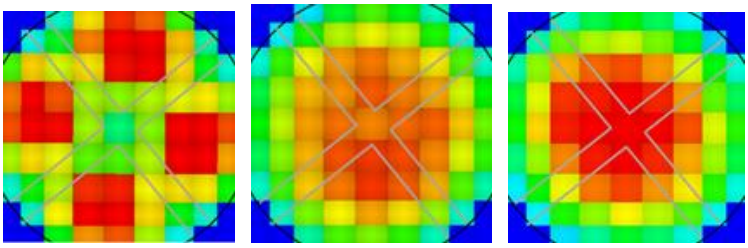


Fig.3 Cross-shaped obstacle

3. NUMERICAL STUDY

In the two-phase flow analysis, the Cartesian coordinate system and staggered grid are used. Surface tension in the momentum equation is estimated by CSF model [3]. Two-phase fluid density is calculated using the densities and the volumetric fractions of both phase.



(a) 20 mm behind (b) 100 mm behind (c) 180 mm behind

Fig.4 Void fraction distributions behind an obstacle

The volumetric fraction, f_m, of liquid or gas phase is calculated by the following conservative equation.

$$\frac{Df_m}{Dt} = -f_m \frac{\partial u_i}{\partial x_i} - \frac{f_m}{\rho_m}\frac{Df_m}{Dt}$$

Where, t, u, x, ρ are velocity, velocity, coordinate and density. Moreover, mass conservation equation of both phases is expressed by

$$\frac{D\rho_m f_m}{Dt} = -\rho_m f_m \frac{\partial u_i}{\partial x_i}$$

Figure 5 shows the predicted result of bubble behavior in a duct with the obstacle. Here, red means the bubble. The geometry is a circular duct with a diameter of 15 mm and length of 50 mm. A cross-shaped obstacle is inserted into the duct. The height and thickness of the cross-shaped obstacle is 5 and 2 mm. 13 air nozzles with a diameter of 1 mm are set to the duct inlet. Small bubbles flow into the duct from the inlet and the bubbles rise vertically toward the downstream. Small bubbles coalesce around obstacles and large bubbles are formed. Figure 6 shows the predicted velocity vector at the vertical center plane of the duct. Since the velocity vectors are disturbed in the vicinity of the obstacle, it can be seen that a complex flow occurs at that position. Depending on the increase of turbulence in the fluid, the bubble shape changes greatly.

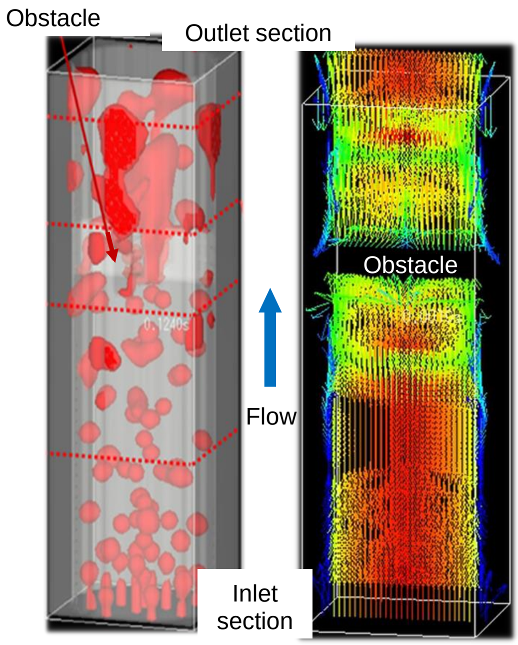


Fig.5 Predicted bubbles in a duct with an obstacle

Fig.6 Predicted velocity vector around an obstacle

4. CONCLUSION

Bubble behavior around the cross-shaped obstacle installed in the vertical circular duct were investigated experimentally and numerically. The void fraction distributions behind the obstacle were measured by using the WMS system and the relationship between the obstacle and the bubble shape was clarified. Resulting from existence of the obstacle and reduction of the duct cross-section, the velocity was disturbed and the bubble shape changed significantly.

4. REFERENCES

1. A. B. Maskal, Mathematical Models of Spacer Grids, ICONE24-60098, USA, (2016).
2. H. M. Prasser, D. Scholz, C. Zippe, Bubble Size Measurement Using Wire-Mesh Sensors, *Flow Measurement and Instrumentation*, 12(4), 299-312 (2001).
3. J. U. Brackbill, D. B. Kothe, C. Zemach, A Continuum Method for Modeling Surface Tension, *J. Computational Physics*, 100, 335-354 (1992).