A Large-Scale Simulation on Water-Vapor Bubbly Flow Dynamics in Fuel Bundles of Advanced Nuclear Reactors

Project Representative
Kazuyuki Takase Japan Atomic Energy Agency

Authors
Kazuyuki Takase *1, Hiroyuki Yoshida *1, Yasuo Ose *2, Takuma Kano *1 and Hajime Akimoto *1
*1 Japan Atomic Energy Agency
*2 Yamato System Engineer Co. Ltd.

In light water reactors each fuel rod is arranged in the shape of a square lattice with an interval of around 3 mm. Several spacers are installed on the surface of the fuel rod with arbitrary axial positions. Water flows vertically along fuel rods and is heated by those, and then many bubbles generate. In order to improve the thermal design procedure of the nuclear reactor core, it is needed to clarify velocity, pressure, temperature and void fraction distributions precisely based on the bubbly flow behavior in a vertical minichannel under the water-vapor two-phase flow condition. However, it is not easy to get those three-dimensional distributions by the experimental study. Then, large-scale two-phase flow simulations were performed to predict the three-dimensional bubby flow configurations in the simply simulated nuclear coolant channel. Regarding both mechanisms of the coalescence and fragmentation of bubbles the useful knowledge was obtained.

Keywords: Fluid dynamics, Large-scale simulation, Two-phase flow, Fuel bundle, Nuclear reactor core, Thermal design

1. Introduction

In light water reactors each fuel rod is arranged in the shape of a square lattice with an interval of about 3 mm. Several spacers are installed on the surface of the fuel rod with arbitrary axial positions. Water flows vertically along fuel rods and is heated by those, and then many bubbles generate. The flow configurations of the liquid-gas two-phase flow change with some parameters such as the mass velocity, channel geometry, flow rate, pressure, heat transfer, etc. These give a large effect to the pressure drop, void fraction, heat transfer and so on. Therefore, in case of conducting the thermal design of the nuclear reactor core, it is requested to clarify the liquid-gas two-phase flow configurations in detail according to the above parameters. To satisfy this request, many two-phase flow experiments using large-scale test facilities have performed, and then a lot of composition equations [1]–[3] which specify the two-phase flow configurations (i.e., bubbly flow, slug flow, annular flow, mist flow, etc.) were proposed based on those experimental data.

Two-phase flow analyses with the two-fluid model codes [4]–[6] have been carried out using the composition equations. Therefore, it is not easy to get high prediction accuracy by using the two-fluid model when experimental data are not enough as an advanced light-water reactor [7], [8]. That is, the two-fluid model is only effective to the average and macroscopic phenomenon in the flow range as the fluid flow characteristic is already clarified. Therefore, it is not the mechanistic numerical method which predicts the unstable interface structure characterizing the liquid-gas two-phase flow behavior.

On the other hand, predicting directly the two-phase flow behavior including complex transient phenomena such as phase change and flow transition without the experimental data, development of a direct two-phase flow simulation method has been performed [9]. Here, "Predicting directly" means that the mathematical models based on the physical phenomena are only used and the composition equations obtained from the experimental data are not used. This paper describes the results of the large-scale numerical simulation on the bubbly flow in a tight-lattice fuel bundle using a newly developed two-phase flow analysis code with the interface tracking method.

2. Advanced Light-Water Reactor

The advanced light-water reactor (RMWR: Reduced Moderation light Water Reactor) [10, 11] which is being developed by Japan Atomic Energy Agency has a higher conversion ratio more than unity by controlling the water flow rates. In order to obtain 1.0 or more conversion ratios, it is expected from the results of the previous studies that a volume ratio of water and fuel must be decreased to about 0.25 or less. To satisfy this condition, the fuel bundle with a triangular tight-lattice arrangement is required: a fuel rod diameter is around 10 mm; and, the gap spacing between each rod is around 1 mm. Although the coolant is 100% liquid water at the core inlet, it changes a mixture of water and vapor along the flow direction, and then, the vapor occupies 90% or more at the core outlet. Therefore, the RMWR has very severe cooling condition on the viewpoint of the thermal engineering.

Figure 1 shows a bird-eye view of the actual RMWR design. It consists of a core, control rod, separator and dryer region, and a pressure vessel. The pressure vessel diameter and height are around 9 and 19 m. The core region is composed of 282 fuel bundles. Each fuel bundle has a hexagonal shape horizontally. A length of one side of a hexagonal shape is about 0.13 m and the axial length of a fuel bundle is about 2.9 m. A heating section in the core consists of two seed and three blanket regions and its length is about 1.3 m (i.e., around 0.2 m in each seed region and 0.3 m in each blanket region). In the core, MOX (mixed oxide) is used to the seed region and then the depleted UO_2 is used to the blanket region.

2. Numerical Analysis

2.1 Analytical Model and Boundary Conditions

Figure 2 shows the present analytical geometry. It simulates a part of the coolant channel of the RMWR and consists of a square duct with a side of 1 mm and an axial length of 25 mm. The hydraulic diameter of this duct is 1 mm and it is almost equivalent to the RMWR condition.

Calculation conditions are as follows:

- Three dimensional duct;
- Dimension of 1 mm × 1 mm × 25 mm;
- Inlet velocity of 0.5 m/s;
- Working fluids of water and air;
- Bubble diameter of 0.1 mm; and,
- Bubble number of 594.

The duct is filled with water first. For 66 axial positions of the duct, the total number of 594 bubbles is put in initially (i.e., 9 bubbles for one axial position). The non-uniform mesh division was applied. The minimum and maximum mesh sizes were 0.001 and 0.1 mm. The number of mesh division in the x, y and z directions are 80 × 80 × 1000, and the total mesh number is around 6,400,000.

On the other hand, boundary conditions are as follows:

- fluid velocities for x, y and z directions are zero on every wall;
- Developed velocity profile is given to the duct inlet; and,
- Periodic boundaries are applied to the duct inlet and outlet for any bubbles.

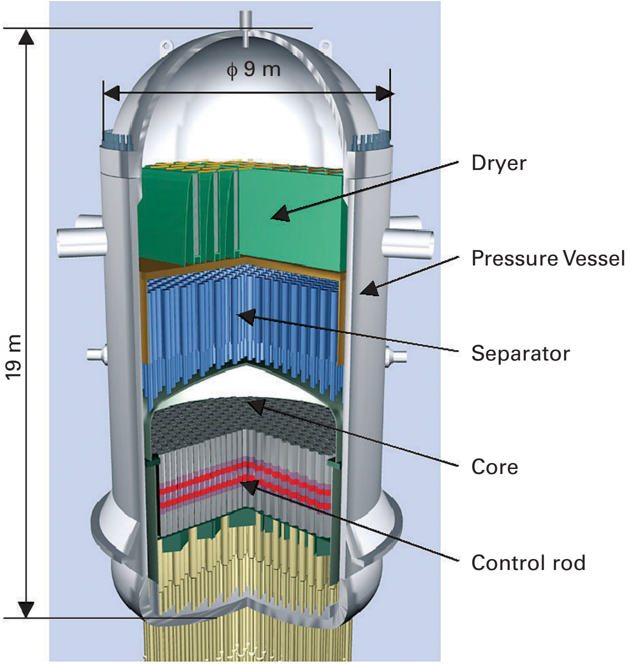


Fig. 1 Structure of the advanced light-water reactor.

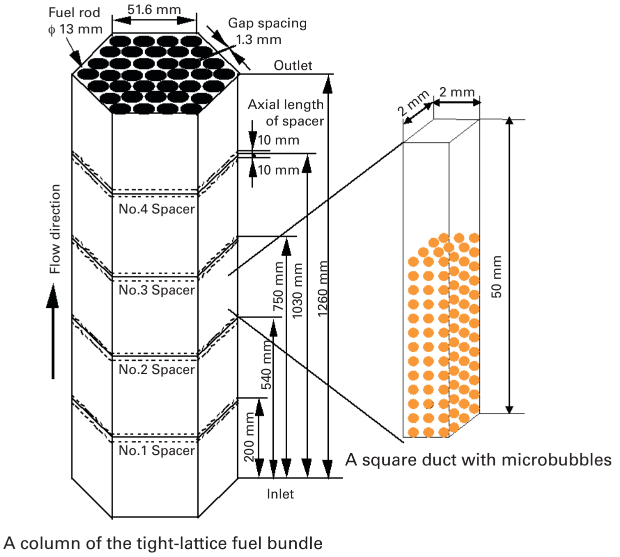


Fig. 2 Analytical geometry of a bubbly flow analysis.

2.2 Computational Ways

The detailed description on the numerical analysis code is already presented by previous ES annual reports. The calculations were performed under the non-heated condition. Although bubbles generally generate by heating, in this calculation the bubbles were given as an initial condition.

The following computational ways were applied:

1) The water flow is calculated by the condition of the initial velocity 0.5 m/s, and a developed velocity profile inside the duct is obtained;
2) Many small bubbles are set to the inside of the duct under the developed velocity condition;
3) A bubbly flow calculation is started;

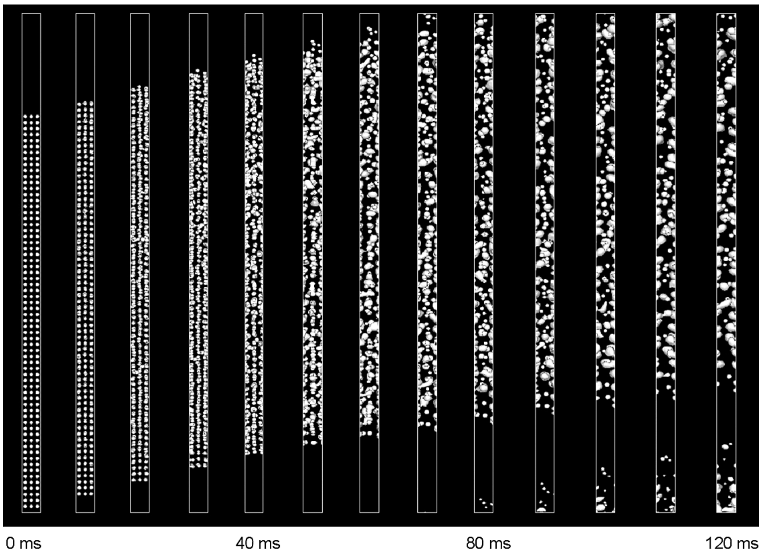


Fig. 3 Predicted bubbly flow configurations.

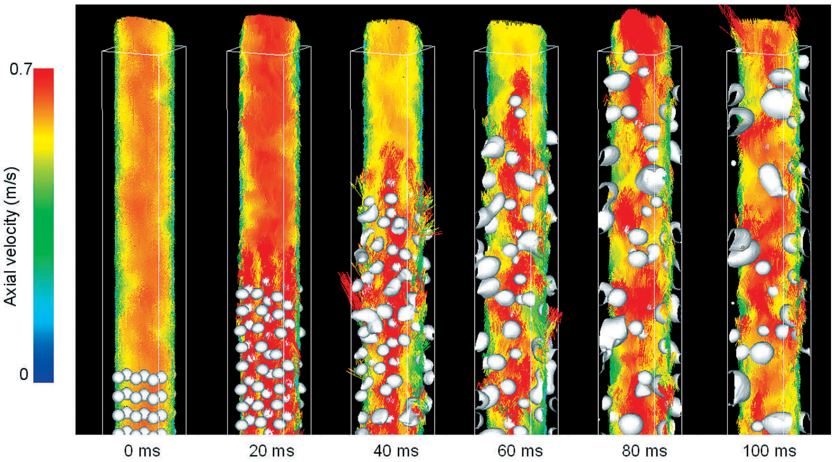


Fig. 4 Predicted axial velocity distributions and bubble motion.

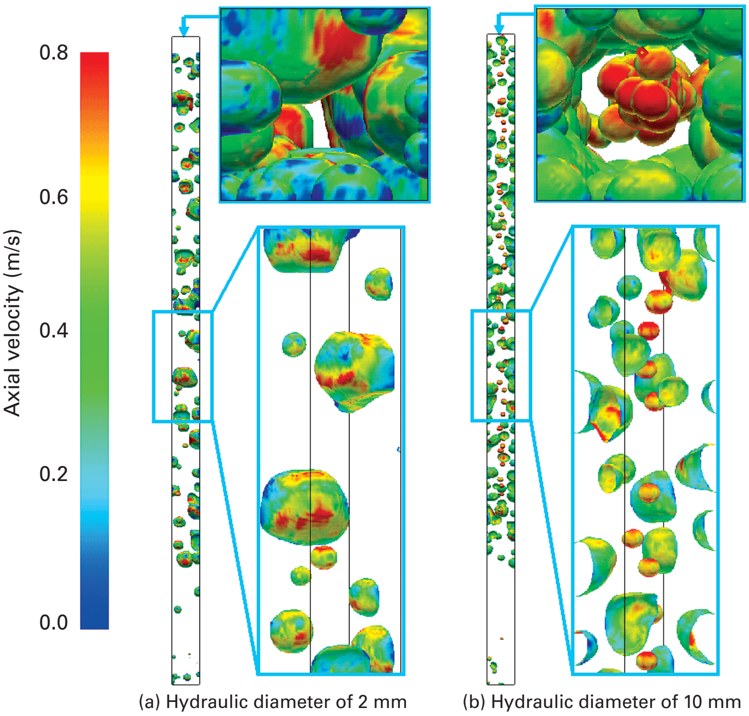
4) Coalescence and fragmentation of bubbles are analyzed; and,
5) The shape of each bubble is repeated at the inlet and outlet of the duct with the cyclic boundary conditions, and bubbly flow calculation is continued.

4. Results and Discussion

Three-dimensional predicted results of the bubbly flow in a square duct are shown in Fig. 3. Water and bubbles flow from a bottom of the duct to a top. Small bubbles are coalesced each other flowing to the outlet of the duct and then gradually large bubbles are formed. An interface between water and bubble changes greatly by coalescence of bubbles. It is promoted by the mixing effect of turbulence generating around each bubble.

Figure 4 shows the axial velocity distributions at the vertical center plane with time. The color contour indicates the axial velocity amplitude: blue is 0 m/s in minimum and red is 0.7 in maximum. The white volume represents one bubble. With progress of time, small bubbles coalescence and a big bubble occurs. According to that, a complicated flow is seen around big bubbles, and the bubble motion becomes dynamic.

Figure 5 shows the relation of the enlargement of micro bubbles and the duct size. Here, Fig. 5(a) is the case that the hydraulic diameter of the square duct is 2 mm, and Fig. 5(b) is the case that it is 10 mm similarly. The color contour shows the axial velocity distribution on the interface of bubble. In Fig. 5(a), coalescence of bubbles is promoted in the center of the duct. Conversely, it is hardly seen near the side wall of the duct. On the other hand, in Fig. 5(b), it is remarkable near the side wall. Moreover, in the center of the duct, it hardly takes place. The tendency that small bubbles go up spirally is seen. For both figures the following tendency is confirmed that the velocity on the interface becomes high near the center of the duct. This is the effect of axial velocity distribution in a core region of the duct.



(a) Hydraulic diameter of 2 mm (b) Hydraulic diameter of 10 mm

Fig. 5 Effect of the hydraulic diameter to the bubble dynamics.

5. Conclusions

For a vertical minichannel simulating the narrow coolant channels in the RMWR core, the large-scale bubbly flow calculations were carried out. The useful knowledge on the coalescence and fragmentation of microbubbles in a minichannel was obtained and the two-phase flow dynamics of the interface between the liquid and gas phase is clarified numerically.

References

[1] Ishii, M., ANL-77-47, Argonne, Illinois, USA, 1977.

[2] Ransom, V. H. et al., NUREG/CR-4312, EGG–2396, 1985.

[3] Wallis, G. B., Trans. ASME, J. Basic Engineering, pp.59–72, 1970.

[4] Kelly, J. E., Kao, S. P., Kazimi, M. S., 1981. THERMIT-2: A two-fluid model for light water reactor subchannel transient analysis. MIT–EL–81–014.

[5] Thurgood, M. J., 1983. COBRA/TRAC - A thermal-hydraulic code for transient analysis of nuclear reactor vessels and primary coolant systems, equation and constitutive models. NURREG/CR-3046, PNL–4385, Vol.1, R4.

[6] Sugawara, S., Miyamoto, Y., 1990. FIDAS: Detailed subchannel analysis code based on the three-fluid and three-field model. Nuclear Engineering and Design, 129, 146–161.

[7] Okubo, T. et al., 2002. Design of small reduced-moderation water reactor (RMWR) with natural circulation cooling. Proceedings of International Conference on the New Frontiers of Nuclear Technology; Reactor Physics, Safety and High-Performance Computing (PHYSOR 2002), Seoul, Korea.

[8] Iwamura, T., 2002a. Core and system design of reduced-moderation water reactor with passive safety features. Proceedings of ICAPP 2002, No.1030, Hollywood, Florida, USA.

[9] Yoshida, H. et al., 2003. Investigation of Water-Vapor Two-Phase Flow Characteristics in a Tight-Lattice Core by Large-Scale Numerical Simulation, Nihon Genshiryoku Gakkai Ronbunshu, Vol.3, No.3, (2004), pp.233–241 (in Japanese).

[10] Iwamura, T. et al., 2002b. Development of reduced-moderation water reactor (RMWR) for sustainable energy supply. Proceedings of The 13th Pacific Basin Nuclear Conference (PBNC 2002), Shenzhen, China, 1631–1637, (2002).

[11] Takase, K. et al., Numerical Analysis of Three-Dimensional Two-Phase Flow Behavior in a Fuel Assembly, Computational Methods in Multiphase Flow III, WIT Press., UK, pp.183–192. (2005).

将来型軽水炉の燃料集合体内気泡流挙動に関する大規模シミュレーション

プロジェクト責任者
高瀬　和之　　日本原子力研究開発機構

著者
高瀬　和之*[1]，吉田　啓之*[1]，小瀬　裕男*[2]，叶野　琢磨*[1]，秋本　　肇*[1]
＊1　日本原子力研究開発機構
＊2　大和システムエンジニア

原子炉熱設計に必要である炉心内水－蒸気系二相流構造の詳細を大規模シミュレーションによって明らかにする研究を行っている。従来の熱設計手法ではサブチャンネル解析コードに代表されるように実験データに基づく構成式や経験式を必要とするが、新型炉に関しては熱流動に関する実験データが十分ではないため、従来手法による熱設計では高精度の予測は困難である。そこで、シミュレーションを主体とした先進的な熱設計手法を開発し、従来手法と組み合わせることによって効率的な新型炉開発の実現を目指している。

本報では、日本原子力研究開発機構が開発を進めている将来型軽水炉の燃料集合体内気泡流に関する大規模シミュレーションの研究成果を示す。将来型軽水炉は、減速材の割合を減らして中性子の減速を抑制することで1以上の高い転換比が期待できる原子炉である。炉心には、直径13mm程の燃料棒が1mm程度の燃料棒間ギャップ幅で三角ピッチ状に稠密に配置される。このような稠密燃料集合体1カラム(1カラムは37本の燃料棒によって構成される)を対象にして3次元シミュレーションを行い、狭隘流路における気泡流挙動を明らかにした。

① 微細な気泡は下流へと移行しながら合体し、次第に大きな気泡が形成される。
② 気泡の合体により気液界面が大きく変形し、それに伴って気泡周囲の流体に複雑な速度場が形成される。
③ 稠密流路体系では、気泡は燃料棒間隔の最も狭い領域よりもそれに隣り合う流路面積の広い領域を流れ易い傾向にある。
④ 隣り合う燃料棒の間隔が狭い領域では水の架橋現象が起こるため、気泡は流れにくい傾向にある。
⑤ 気泡の運動は流れ方向に対する移動が支配的であり、クロスフローのような水平断面方向への移動は小さい傾向にある。

キーワード：熱流動，大規模シミュレーション，二相流，気泡挙動，稠密燃料集合体，新型炉熱設計