Large-Scale Numerical Simulations on Thermal-Hydraulics of Multi-Phase Flows in Nuclear Reactors with a Direct Analysis Method

Group Representative
Kazuyuki Takase Japan Atomic Energy Research Institute, Principal Research Scientist

Authors
Kazuyuki Takase Japan Atomic Energy Research Institute, Principal Research Scientist
Hiroyuki Yoshida Japan Atomic Energy Research Institute, Research Scientist
Yasuo Ose Yamato System Engineer, Research Scientist

A reduced-moderation light water reactor (RMWR), which is studied by the Japan Atomic Energy Research Institute, needs to the volume ratio between the coolant and fuel less than 0.2, in order to achieve 1.1 or more high breeding ratios. The RMWR fuel assembly consists of many fuel rods with a diameter of around 10 mm with the triangular tight-lattice configuration in the radial direction. A minimum gap width between fuel rods is about 1 mm. Moreover, spacers are set to keep uniformly the gap width between fuel rods for horizontally. These spacers are installed with a fixed interval to the flow direction in a reactor core. As a result of this, a complicated flow is formed around each spacer because of a large flow disturbance. Then, fundamental liquid film flow behavior in a narrow channel with a three-dimensional rectangular rib was analyzed numerically using a newly developed two-phase flow analysis code. From this study the following conclusions were derived: a wake behind the spacer rib takes place easier when the initial film thickness is short and then the gas velocity is higher than the liquid velocity; and, the liquid film thickness reduces with increasing the gas velocity.

Keywords: Reduced-Moderation Light Water Reactor, Liquid Film, Spacer Rib, Wake, Flow Separation.

1. Introduction

A reduced-moderation light water reactor (RMWR), which is studied by the Japan Atomic Energy Research Institute, needs to the volume ratio of the coolant and fuel less than 0.2, in order to attain 1.1 or more high breeding ratios. The RMWR fuel assembly consists of many fuel rods with a diameter of around 10 mm with the triangular tight-lattice configuration in the horizontal direction. Here, a minimum gap width between fuel rods is about 1 mm. Moreover, spacers are set to keep uniformly the gap width between fuel rods for horizontally. These spacers are installed with a fixed interval to the flow direction in a reactor core. As a result of this, a complicated flow is formed around each spacer because of a large flow disturbance.

Many studies on the thermal-hydraulic characteristics around spacers and obstacles in narrow channels have been conducted for a field of the single-phase flow. On the other hand, as for the thermal-hydraulics in the liquid film flows inside narrow channels with spacer ribs, the flow visualization experiments and the measurements of the liquid film thickness on ribbed channels have been carried out. However, shapes and geometries of spacer ribs in cases of previous studies are different from those in case of the present study. Therefore, it is not appropriate from the results of the previous research to predict the two-phase flow behavior in narrow channels with spacer ribs. Then, the present study was conducted to investigate numerically the fundamental liquid film flow behavior in a narrow rectangular channel with a three-dimensional spacer rib. This paper describes analytical results of a water-vapor two-phase film flow on a ribbed surface.

2. Numerical Analysis

The two-phase flow analysis code TPFIT developed by Yoshida was used for the present numerical analysis. This code can transport an interface between the liquid and gas in the time and space directions with high accuracy. The TPFIT is discretized using the CIP method based on the non-conservative fluid equations. In the CIP method, the density function in an interface is assumed to be continuous. With regard to this density function, the interface tracking is carried out by the digitizer function (i.e., tangent conversion). When the density function in an interface does not have consistency with the digitizer function or the small bubbles and droplets generate, therefore, there is a possibility that a part of bubbles and droplets may disappear. This is due to that the interface tracking procedure is not established enough yet. Then, Yoshida improved as a transportation method of the volume of fluid. As a result of this, a practical two-phase flow analysis with the CIP method was available.

3. Analytical Model and Boundary Conditions

The analytical model consists of a three-dimensional rectangular channel with a simplified spacer rib. Since the channel height is equal to the rib height, the flow does not overcome the spacer rib. It is thought that the fluid flow from the channel inlet separates right and left at the front of the spacer rib and then flows back. Dimensions of length and width at the channel are 81 and 27 mm and those at the spacer rib are 6 and 3 mm. The height (H) is changed in the range of 2-10 mm as a parameter. Boundary conditions are as follows: top and bottom at the channel and the surface of the spacer rib are walls; every wall is the no slop condition; both sides at the channel are set to the symmetric boundary and free slip conditions; and the velocity profile at the channel inlet is uniform.

Water and vapor are used in the present analysis. The water flows near the bottom wall at the channel and the vapor flows near the top wall. In the calculations the film thickness (h), water velocity (J_l) and vapor velocity (J_g) were given to the channel inlet as initial values. The film flow behavior around the spacer rib progressing in the time direction was analyzed numerically changing the channel and spacer rib height, vapor velocity and film thickness. The calculations were carried out in the range of the following conditions: non-heated isothermal flow; h=1-7 mm; J_l=1 m/s; and J_g=1-10 m/s. Thermal properties of water and vapor were calculated using a saturation temperature 288°C at pressure 7.2 MPa. These values simulate the coolant condition at the vicinity of the RMWR core outlet.

4. Results and Discussion

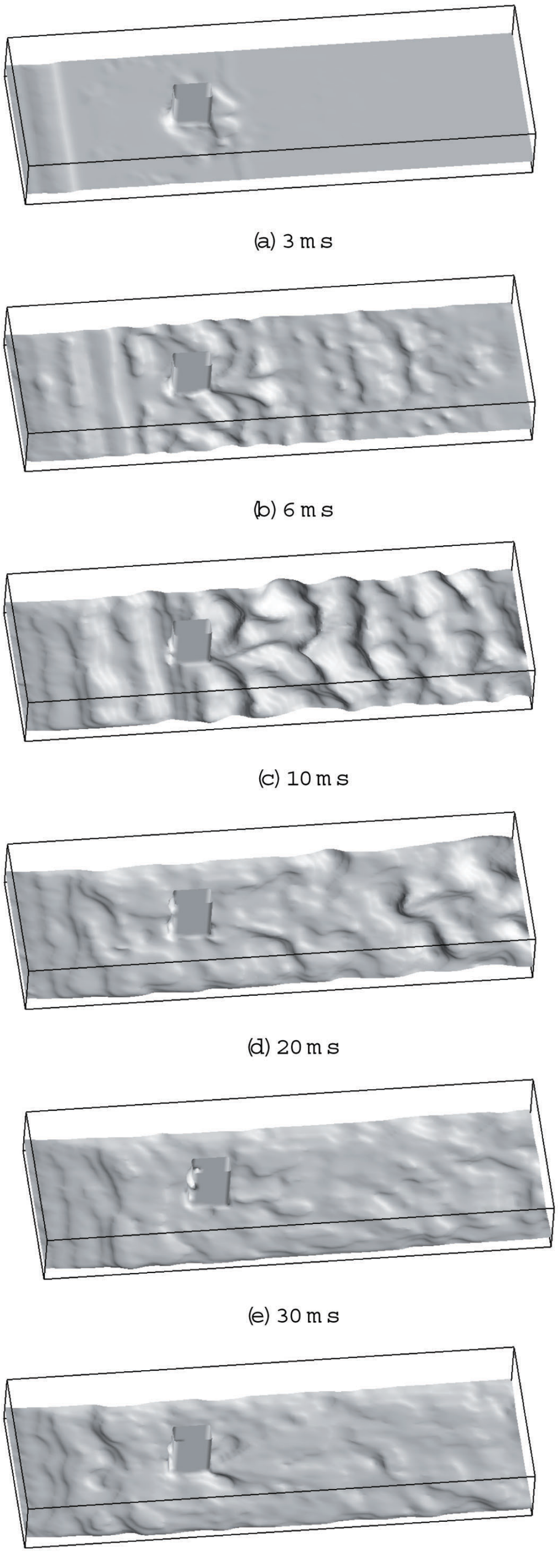
Figure 1 shows the visualized interface behavior around a spacer rib. Each interface corresponds to the void fraction of 0.5. Calculated conditions are as follows: H=10 mm, h=3 mm, J_l=1 m/s, and J_g=5 m/s. A large flow disturbance takes place around the spacer rib with the start of the calculation (Fig. 1 (a)). It is amplified with time and changes a wavy flow and then advances back (Fig. 1 (b), (c)). The wavy flow disappears gradually with decreasing the flow instability (Fig. 1 (d) (e)). A short separated line behind the spacer rib is seen. (Fig. 1 (f))

Similarly, the visualized interface behavior around a spacer rib at H=10 mm, h=3 mm, J_l=1 m/s, and J_g=10 m/s is shown in Fig. 2. The J_g is twice as high as that in Fig. 3. Behavior on the flow disturbance and waving is almost the same as the case of Fig. 1 (Fig. 2 (a), (b), (c)). Since J_g is faster than that in Fig. 1, it is thought that an interface receives a strong shear stress in comparison with the case in Fig. 1. Consequently, it was confirmed that a part of the interface is broken and takes off (Fig. 2 (d)). Along separated lines generating from the rear end of the spacer rib, a wake is formed (Fig. 2 (e)). In the wake region, the liquid film thickness becomes extremely thin due to a large flow disturbance and is mostly filled with vapor (Fig. 2 (f)). From this result, if the spacer ribs are installed in a line with an arbitrary interval for the flow direction, the spacer rib located downward will receive the effect of a wake by the spacer rib located upward. Therefore, it is very important to consider the optimum spacer rib arrangement.

5. Conclusions

Three-dimensional computations on a liquid film flow around a simulated spacer rib under a simplified RMWR flow channel condition carried out using a newly developed two-phase flow analysis code TPFIT. A fundamental effect of a spacer rib on the liquid film flow was clarified numerically and the following conclusions were derived.

1) A wake behind the spacer rib takes place easier when the initial liquid film thickness is small and then the gas velocity is higher than the liquid velocity.
2) The liquid film thickness reduces with increasing the gas velocity and with decreasing the channel height.


(a) 3 ms

(b) 6 ms

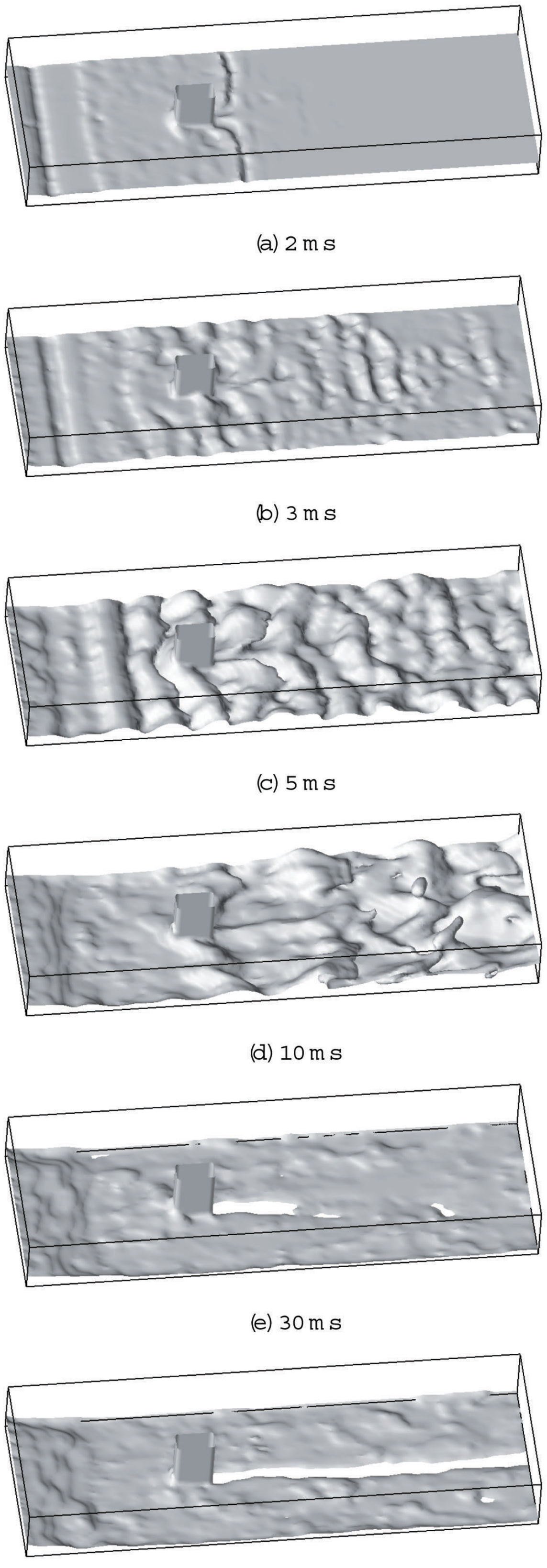
(c) 10 ms

(d) 20 ms

(e) 30 ms

(f) 100 ms

Fig. 1 Predicted film flow patterns with time from the start of the calculation at H=10 mm, h=3 mm, J_l=1 m/s and J_g=5 m/s.


(a) 2 ms

(b) 3 ms

(c) 5 ms

(d) 10 ms

(e) 30 ms

(f) 40 ms

Fig. 2 Predicted film flow patterns with time from the start of the calculation at H=10 mm, h=3 mm, J_l=1 m/s and J_g=10 m/s.

熱流動直接数値解析手法による原子炉内混相流に関する大規模シミュレーション

利用責任者
高瀬　和之　　日本原子力研究所　主任研究員

著者
高瀬　和之　　日本原子力研究所　主任研究員
吉田　啓之　　日本原子力研究所　研究員
小瀬　裕男　　大和システムエンジニア　研究員

低減速軽水炉の炉心燃料集合体内には、1mm程度の燃料棒間ギャップを一定に保つためにスペーサが設置される。このような狭隘流路に存在するスペーサ周辺の熱流動特性に関しては、単相流では多くの研究が行われているが、二相流ではほとんど見られない。そこで、狭隘流路内に置かれたスペーサ等の物体が二相流挙動に及ぼす影響を地球シミュレータを利用した大規模シミュレーションによって調べた。数値解析には吉田が開発した二相流コードTPFITを使用した。解析体系は3次元流路とスペーサ形状を簡略模擬した矩形突起から成る。解析では、流路入口に液膜厚さとその流速及び蒸気流速を与え、時間方向に進展する液膜流挙動を非加熱等温流条件に対して定量的に調べた。計算に使用した入力値は低減速軽水炉の炉心出口近傍の条件を模擬した。解析の結果、突起後端から発生するはく離線に沿ってウエークが形成され、ここでは強い乱れによって液膜が排除され、ほぼ蒸気で満たされることがわかった。このことから、流れ方向に突起を一定間隔で設置する場合には、前方の突起で発生したウエークが後方の突起に順次影響するため、突起の配置の検討が重要であることが推察された。

キーワード：低減速軽水炉、液膜流解析、3次元流路、スペーサ、ウエーク、はく離、突起配置