Large-Scale Simulations on Thermal-Hydraulics in Fuel Bundles of Advanced Nuclear Reactors

Project Representative
Kazuyuki Takase Japan Atomic Energy Agency

Authors
Kazuyuki Takase [*1], Hiroyuki Yoshida [*1], Yasuo Ose [*1], Takuma Kano [*1], Elia Merzari [*2] and Hisashi Ninokata [*2]

*1 Japan Atomic Energy Agency
*2 Tokyo Institute of Technology

In order to predict the water-vapor two-phase flow dynamics in a fuel bundle of an advanced light-water reactor, large-scale numerical simulations were performed using a highly parallel-vector supercomputer, the earth simulator. Although conventional analysis methods such as subchannel codes and system analysis codes need composition equations based on the experimental data, it is difficult to obtain high prediction accuracy when experimental data are nothing. Then, the present large-scale direct two-phase flow simulation method was proposed, and the void fraction distributions under the heated flow condition were analyzed. On the other hand, in order to estimate the turbulence characteristics in the fast breeder nuclear rector core, large eddy simulation and direct numerical simulation were carried out under the concentric and eccentric annular channels simulating the simplified bare fuel bundle. An objective of this work is to identify reliable and practical approaches for the simulation of unsteady flows in rod-bundles and related geometries.

Keywords: Large-scale simulation, Thermal hydraulics, Fuel bundle, LES, DNS, Turbulence, Nuclear reactor

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. 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 predicted results of three-dimensional void fraction in a tight-lattice fuel bundle under a heated flow condition.

Moreover, Computational Fluid Dynamics (CFD) techniques in recent years have been successfully applied to the simulation of fuel bundles and other components of the nuclear reactor. Generally, calculation codes are based on the Reynolds Averaged Navier-Stokes (RANS) approach and the most of all calculations are usually performed a steady state condition. In order to simulate turbulent flows, conventional turbulence modeling for the RANS equations is not always reliable. Even in simple flows (i.e. flows in fuel bundles) the results can be not accurate when particular conditions occur; for examples, buoyancy, flow oscillations, and turbulent mixing. A proper methodology under these conditions was developed, in order to predict detailed temperature distributions in extreme design or accident conditions. Traditionally, two approaches are employed for the simulation of full transients; the direct simulation of the Navier-Stokes equations (DNS) and Large Eddy Simulation (LES).

This research theme consists of two projects; a large-scale two-phase flow simulation by Japan Atomic Energy Agency and a large-scale turbulent flow simulation by Tokyo Institute of Technology. The result of the former is shown in Section 2 and that of the latter is shown in Section 3.

2. Void Fraction Distributions in Fuel Bundles of Advanced Nuclear Reactors

2.1 Advanced Light-Water Reactor

The advanced light-water reactor of Japan Atomic Energy Agency [10] has a higher conversion ratio more than unity by controlling the water flow rates. In order to obtain 1 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% 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 advanced light-water reactor has very severe cooling condition on the viewpoint of the thermal engineering.

Figure 2.1.1 shows a bird-eye view of the actual advanced light-water reactor 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.

2.2 Numerical Analysis and Typical Result [11]

Figure 2.2.1 shows the analytical geometry consisting of 37 RMWR fuel rods. The geometry and dimensions simulate the experimental conditions. Here, the fuel rod outer diameter is 13 mm and the gap spacing between each rod is 1.3 mm. The casing has a hexagonal cross section and a length of one hexagonal side is 51.6 mm. An axial length of the fuel bundle is 1260 mm. The water flows upward from the bottom of the fuel bundle. A flow area is a region in which deducted the cross-sectional area of all fuel rods from the hexagonal flow

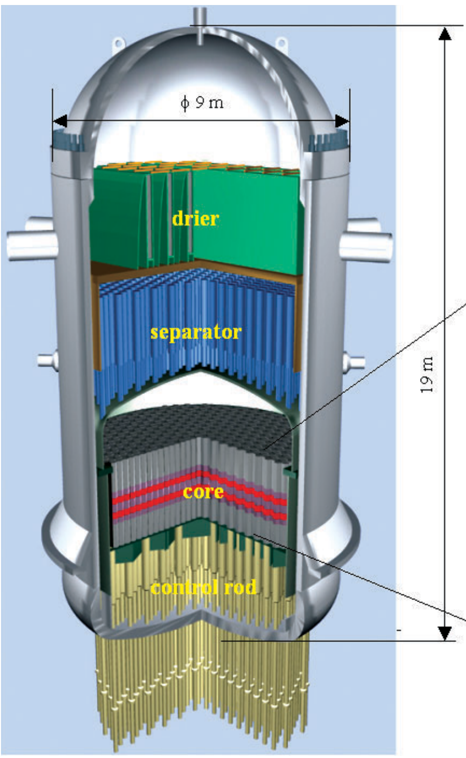

Fig.2.1.1 A bird-eye view of the advanced light-water reactor.

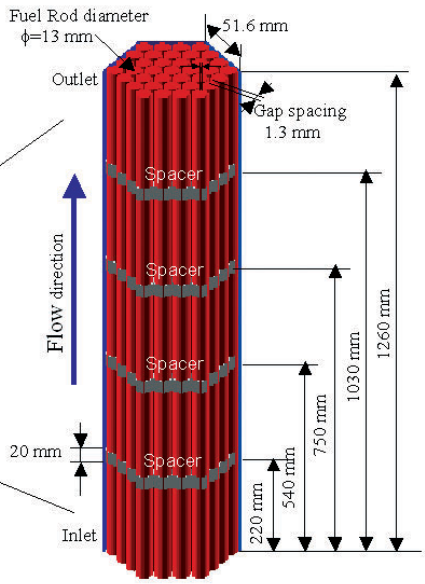

Fig.2.2.1 Analytical geometry of a tight-lattice fuel bundle.

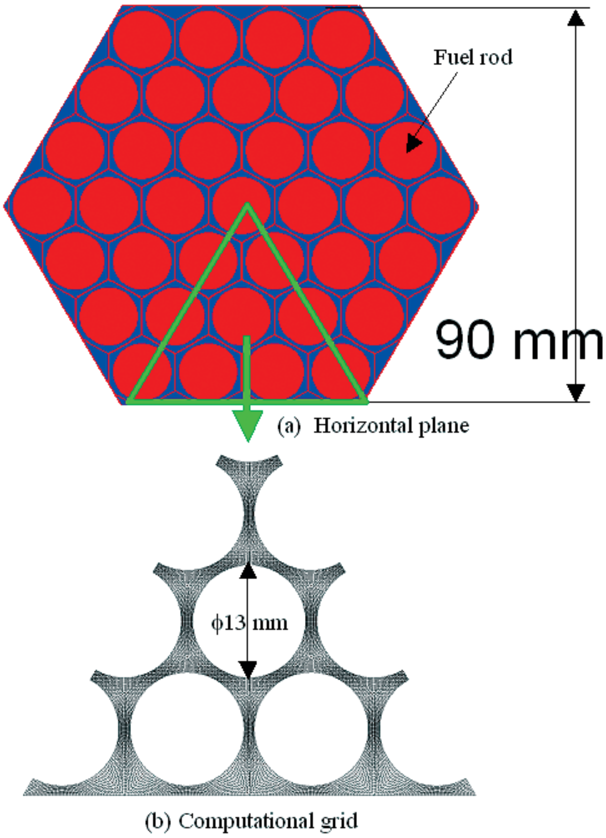

Fig.2.2.2 Computational grid condition.

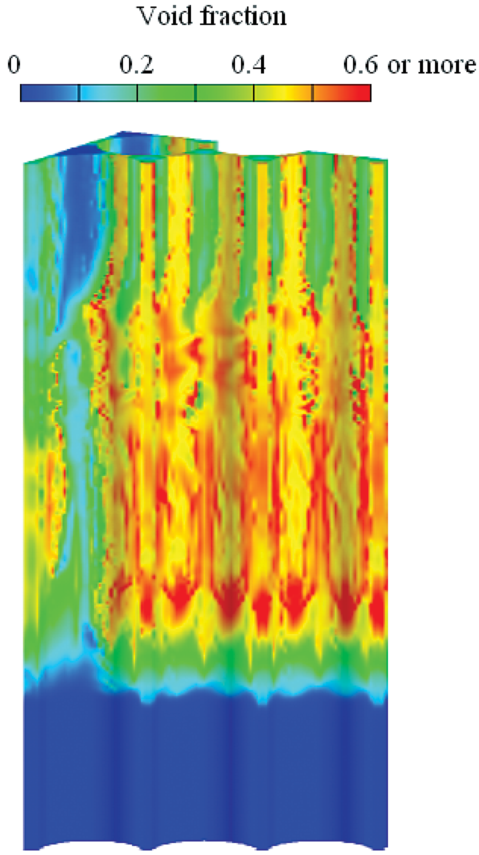

Fig.2.2.3 Predicted three-dimensional void fraction dostribution.

passage. The spacers are installed into the fuel bundle at the axial positions of 220, 540, 750 and 1030 mm from the bottom. The axial length of each spacer is 20 mm.

Figure 2.2.2 shows the present computational grids, which corresponds to one sixth of a horizontal plane. A non-uniform mesh division was applied. The total number of mesh division in the x, y and z directions are 20 million. Here, boundary conditions are as follows:

1) Fluid velocities for x, y and z directions are zero on every wall;
2) Developed velocity profile is given to the duct inlet; and,
3) Heat flux of each fuel rod was given to the heating section.

A three-dimensional predicted result of void fraction in a fuel bundle is shown in Fig.2.2.3. The color contour indicates the void fraction distribution; blue is the liquid water (i.e., void fraction is 0) and red is the mixture of water and vapor (void fraction more than 0.6). The boiling occurs at the heated section which is positioned at the center for vertically. Although the coolant is the liquid water at the inlet section of the fuel bundle, it changes water and vapor due to the boiling by fuel rods. The void fraction near wall region is lower than the center region in the radial direction because the heat transfer rate at the near wall region is lower than that at the center region.

3. Direct Numerical Simulation and Large Eddy Simulation of Fluid Flow Characteristics in Simulated Fuel Bundle

3.1 Geometry and Mesh

The geometries studied in the present work (see Fig. 3.1.1) include:

1) An eccentric channel with an eccentricity equal to 0.5 and a hydraulic diameter equal to 1.
2) An eccentric channel with an eccentricity equal to 0.95 and a hydraulic diameter equal to 1.
3) A concentric channel with a parameter α equal to 0.1 and a hydraulic diameter equal to 0.9.

Experimental data and calculation results related to the two of the layouts can be found in Nouri et al. [12] and Seo et al. [13]. The parameter α is defined as:

$$\alpha = D_{out} / D_{in} \tag{3.1.1}$$

The value of α influences the maximum value of the streamwise velocity as well as the position where the turbulent shear stress vanishes. More generally the local deformation of the velocity profile compared to a parallel plate simulation can be expressed as a function of the Reynolds number and α. In the case of eccentric channels another parameter influences the shape of the flow. Let us define d as the distance between the two cylinders axis (we will assume the two axis to be parallel and the cylinders infinite). The eccentricity is therefore defined as:

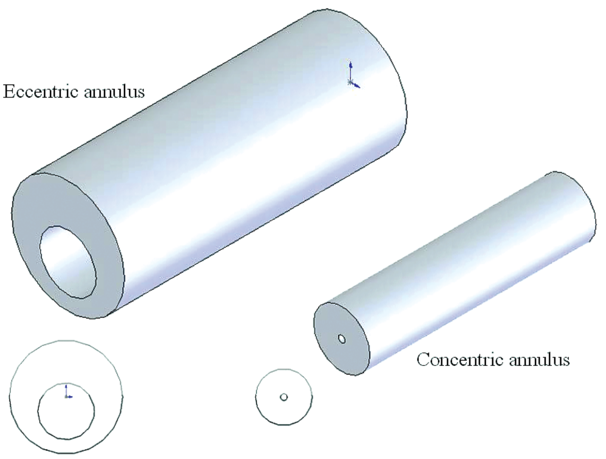

Fig.3.1.1 Geometries considered in the present calculation.

$$e = d / (D_{out} - D_{in}) \tag{3.1.2}$$

And its value is a real number included between 0 (concentric channel) and 1 (the two cylinders touch in at least one point). Since our aim is to study fully developed turbulent flows periodic boundary conditions in the streamwise direction have been selected. The length of the domain in the streamwise direction has been therefore chosen coherently with the choice of the boundary condition, following the concept of minimal flow unit [14]. The length has been selected so that the two point correlation tensor between points at half-length distance to be negligible.

3.2 Numerical Practices and Typical Results

The codes employed present the following features:

1) the incompressible Navier-Stokes equations have been solved in boundary fitted coordinates;
2) a fractional step algorithm has been employed;
3) a second order spatial discretization scheme;
4) has been adopted, for all terms involved;
5) during the simulation the CFL (Courant-Friedrichs-Lewy) has always kept below 0.1;
6) an explicit time advancement trough the Adams-Bansforth scheme has been implemented; and,
7) Poisson equation has been solved using a pseudo-spectral method in the stream-wise direction and a conjugate gradient solver in the other directions.

In the case of LES, a dynamic model eddy viscosity model with Lagrangian integration [15] has been used. The particular choice of the numerical method for the pressure equation permits a high level of parallelization. As for the vectorization, the code has been thoroughly modified to ensure it uses at its best the characteristics of the Earth Simulator system.

This work is, from many points of view, an extension and development of the results carried on [16]. The Reynolds range has been extended in order to reach values of the Reynolds number for which extended experimental data are available. Moreover different values of the eccentricity ratio have also been explored. In Table 2.3.1 are reported the cases run in Earth Simulator. Case B employed more than 176 million meshes, and took 6 months to reach agreeable results.

As typical results of present work, three dimensional distributions of the streamwise velocity for cases C and B are shown Figs. 2.3.1 and 2.3.2, respectively. In Fig. 2.3.1, the instantaneous velocity distribution for case C is presented and in Fig. 2.3.2 the averaged distribution for the streamwise velocity for case B (over 3 s/ 300.000 time steps and along the streamwise direction) is presented.

4. Conclusion

In the present work, the following conclusions were derived:

1) In order to predict the water-vapor two-phase flow dynamics in the RMWR fuel bundle and to reflect them to the thermal design of the RMWR core, a large-scale simulation was performed under a full bundle size condition using the earth simulator. Details of water and vapor distributions under the heated condition were clarified numerically.

Table 2.3.1 DNS and LES calculation cases.

Case	Reynolds number	Grid number	Nodes	Eccentricity	Run time	DNS / LES
A	8000	256x100x336	1	0.95	3 months	DNS
B	26600	768x300x768	16	0.5	6 months	DNS
C	8900	128x64x128	1	0	1 month	LES
D	13300	256x80x128	1	0.5	1 month	LES

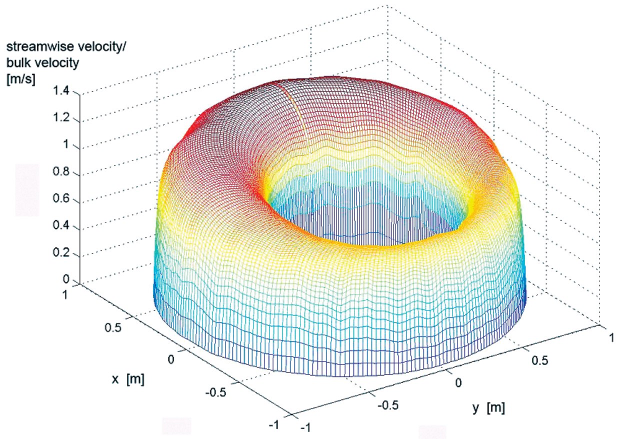

Fig.2.3.1 Three-dimensional plot of the averaged streamwise velocity at case B.

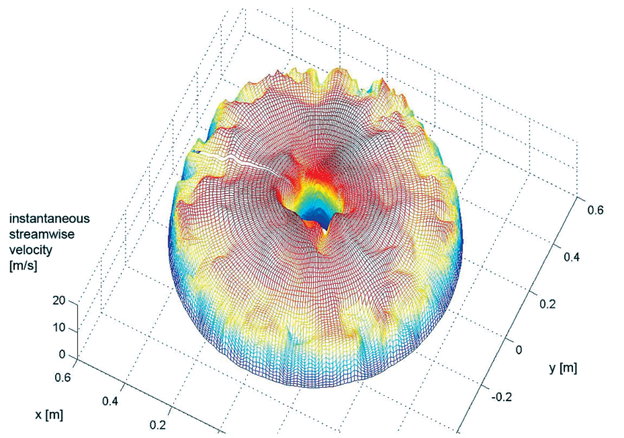

Fig.2.3.2 Three-dimensional plot of the instantaneous streamwise velocity at case C.

2) LES and DNS of the flow in annular channels have been presented. Substantial agreement with the experimental data has been obtained. Experimental observations have been confirmed. The DNS data has been used to thoroughly test LES models in these geometries, and identify a practical methodology for the simulation of the flow in rod-bundles [17].

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., et al., THERMIT-2: A two-fluid model for light water reactor subchannel transient analysis, MIT-EL–81–014, (1981).

[5] Thurgood, M. J., 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, (1983).

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

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

[8] Iwamura, T., Core and system design of reduced-moderation water reactor with passive safety features, International Congress on Advances in Nuclear Power Plants, No.1030, Florida, USA (2002).

[9] Yoshida, H. et al., 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, pp.233–241, (2004) (in Japanese).

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

[11] Takase, K. et al., A large Scale Simulation of Two Phase Flow Characteristics around Fuel Rods in a Tight-Lattice Core, ASME International Mechanical Engineering Congress, Chicago, USA, 2006.

[12] Nouri, J. M., et al., Flow of newtonian and non-newtonian fluids in concentric and eccentric annuli, Journal of Fluid Mechanics, Vol. 253, pp.617–64, (1993).

[13] Chung, S. Y., et al., Direct numerical simulation of turbulent concentric annular pipe flow Part 1: Flow field, International Journal of Heat and Fluid Flow, Vol.23, pp.426–440, (2002).

[14] Jiménez, J., et al., The minimal flow unit in near-wall turbulence, Journal of Fluid Mechanics, Vol.225, pp.213–240, (1991).

[15] Meneveau, C., et al., A lagrangian dynamic subgrid-scale model of turbulence, Journal of Fluid Mechanics, Vol.319, pp.353–385, (1996).

[16] Okumura, T., et al., Direct numerical simulation of the flow in annular channels, ANS Winter Meeting 2006, Albuquerque, USA, (2006).

[17] Merzari, E., et al., Test of LES SGS models for the flow in annular channels, International Congress on Advances in nuclear Power Plants, Nice, France, (2007).

将来炉用燃料集合体内の熱流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*1，叶野　琢磨*1，Elia Merzari*2，二ノ方　壽*2
＊1　日本原子力研究開発機構
＊2　東京工業大学

原子炉内の熱流動挙動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、将来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将来炉開発の実現を目指している。

本プロジェクトは2つの研究テーマから成る。1つは将来型軽水炉の燃料集合体内熱流動挙動を大規模シミュレーションによって計算機上に再現する研究であり、日本原子力研究開発機構が担当している。もう1つは、高速炉の炉心燃料チャンネル内乱流挙動を大規模シミュレーションによって解明する研究であり、これは東京工業大学が行なっている。

本報では、日本原子力研究開発機構が行なった稠密燃料集合体内水—蒸気二相流挙動の詳細予測結果について報告する。また、東京工業大学が行なった大規模乱流シミュレーションの結果について報告する。

キーワード：大規模計算, 熱流動, 燃料集合体, 乱流解析, 原子炉