CFD Analysis on the Prismatic Fuel Assembly of a Very High Temperature Reactor

Nam-il Tak, Min-Hwan Kim, Won-Jae Lee
KAERI, 1045 Daedeok Street, Yuseong-gu, Daejeon 305-353, Korea, takni@kaeri.re.kr

1. Introduction

The maximum fuel temperature under steady-state operating conditions is the one of the main parameters which are considered as the design limits of a very high temperature reactor (VHTR). In the past, computational fluid dynamics (CFD) analyses were made on a typical unit cell in the nuclear fuel arrangement to obtain the maximum hot spot fuel temperature [1,2]. The unit cell approach is a very economical way to reduce the computational efforts. In a prismatic type of a VHTR, however, the unit cell approach cannot consider a heat transfer within a fuel assembly as well as a coolant flow through a gap between the fuel assembly blocks, which may affect the maximum fuel temperature.

In this work, three-dimensional CFD analysis is carried out on the standard fuel assembly of PMR600 (GT-MHR [3] with a helium exit temperature of 950°C) in order to investigate the temperature distribution within the fuel assembly and to assess the accuracy of the unit cell approach.

2. Physical Model

Fig. 1 shows the geometry of the standard fuel assembly of GT-MHR. The hexagonal graphite block has 210 blind holes for fuel compacts and 108 channels for a helium coolant flow. The height of one fuel block is 79.3 cm. Since 10 fuel blocks are stacked and form a column in the active core, the height of the active core is 793 cm. At both the top and bottom of the active core, graphite reflector blocks are placed. The hexagonal blocks are doweled together to align the coolant holes between the stacked blocks.

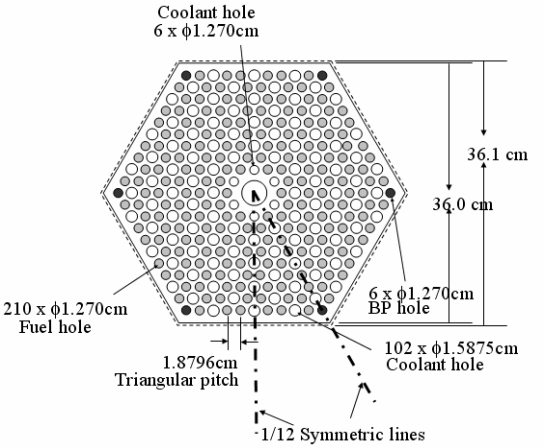

Fig. 1. The geometry of the standard fuel assembly of GT-MHR (4 dowel holes are not shown.).

A cold helium flows downward through the coolant holes to remove the heat generated in the fuel compacts.

3. Numerical Analysis

Table 1 shows the main thermo-fluid conditions for the present CFD analysis.

Table 1. The main thermo-fluid conditions

Assembly power (MW)	5.88
Assembly flow rate (kg/s)	2.46 kg/s
Inlet He temperature (°C)	490
Ave. He outlet temperature (°C)	950

By assuming small effects of the 4 dowel holes, 1/12 part of the fuel assembly is considered for the CFD analysis (See Fig. 1). A uniform power density is applied in the fuel compact. It is further assumed that the gap size between the assemblies is 1 mm and 1.1% of the coolant flow is bypassed through the gap.

The CFD calculation is performed by using a commercial code CFX 11 [4]. The standard k-ε turbulence model with the scalable wall function is applied to the main coolant flows and the bypass flow through the gap is assumed to be laminar.

4. Numerical Results

Fig. 2 shows the temperature contour calculated by CFX 11 on the maximum temperature plane.

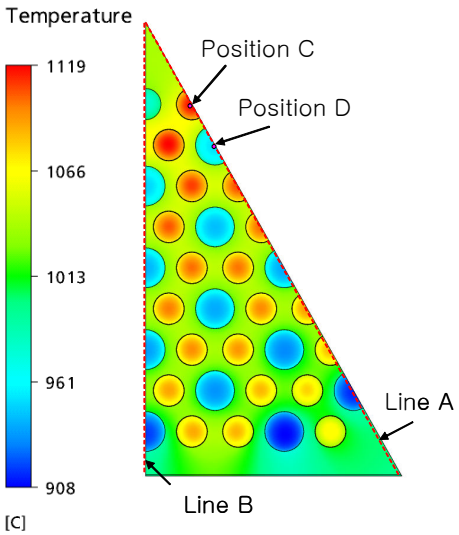

Fig. 2. Temperature contour on the maximum temperature plane.

The maximum fuel temperature is predicted as high as 1119 °C and is located at the innermost fuel. The lowest coolant temperature is predicted at the outmost region. Fig. 3 shows the temperature distributions along the two specified lines in Fig. 2 (i.e., Line A & Line B). The largest temperature gradients are seen at the boundary layers of the coolant flows. The maximum temperature difference between the fuel holes is predicted as 34°C and the predicted maximum temperature difference between the coolant holes is 65°C.

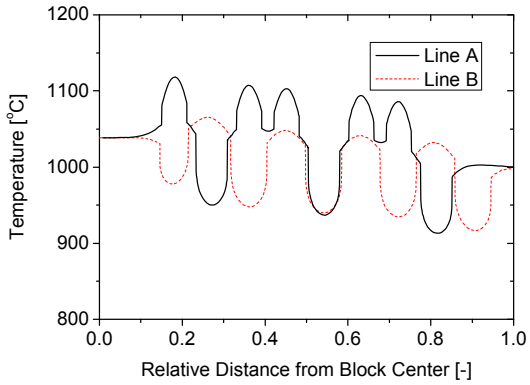

Fig. 3. Temperature distributions along Line A and B.

In order to examine the vertical temperature profiles at the active core, the temperature profiles are plotted along the height of the two specified positions in Fig. 2 (i.e., Position C & Position D). As expected, the coolant temperature is linearly increased with the flow direction. The sharp temperature drops at the fuel are due to the graphite plugs between the stacked fuel blocks.

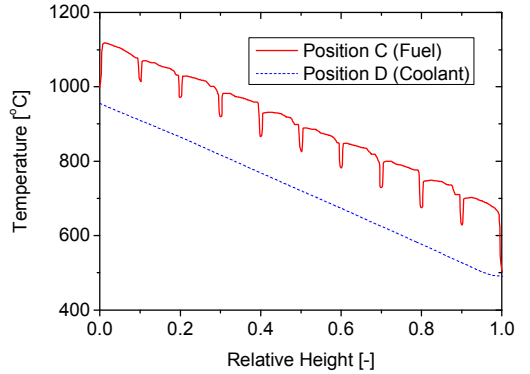

Fig. 4. Vertical temperature distributions at Position C and D.

Table 2 shows the comparison of the results by the unit cell model with those by the present model. It clearly shows that the overall predictions by the unit cell approach are reasonable. It should be noted, however, that the unit cell approach does not provide conservative results. The predicted maximum fuel temperature by the unit cell model is lower by 20°C.

Table 2. Comparison of the results by the unit cell model with those by the present model

	Unit Cell	1/12 Fuel Assembly
Max. coolant velocity (m/s)	53.9	50.8
Active height pressure drop (kPa)	24.4	25.2
Ave. fuel temperature (°C)	865	868
Max. fuel temperature (°C)	1099	1119

5. Conclusions

Three-dimensional CFD analysis has been made on a standard fuel assembly of PMR600 including a bypass flow through a gap. The results of the CFX show significant temperature variations within the fuel assembly although a uniform power profile is applied. It also shows that the unit cell approach can be a reasonable approximation for the real situations. For an evaluation of the final design of a fuel assembly, however, CFX analyses on the fuel assembly including the bypass flow are essentially required to assess the thermal margin accurately.

ACKNOWLEDGMENTS

This work was financially supported by the Korean Ministry of Science and Technology.

REFERENCES

[1] N. I. Tak et al., “Thermal Hydraulic Assessment of Two Types of Prismatic Fuel Assemblies,” Transactions of the Korean Nuclear Society Autumn Meeting, Busan, Korea, Oct. 2005.
[2] W. K. In et al., “Three-Dimensional Analysis of the Hot-Spot Fuel Temperature in Pebble Bed and Prismatic Modular Reactors,” Transactions of the Korean Nuclear Society Spring Meeting, Chuncheon, Korea, May 25-26, 2006.
[3] General Atomics, GT-MHR Conceptual Design Description Report, Part I. Nuclear Regulatory Commission, August 2002.
[4] ANSYS Incorporated, CFX 11 Manuals, 2006.