Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 10-11, 2007

Numerical Analysis of Liquid Metal Cooled Rod Assembly

Hyoung M. Son, Kune Y. Suh
Seoul National University, San 56-1 Sillim-dong, Gwanak-gu, Seoul, 151-744, kysuh@snu.ac.kr

1. Introduction

A benchmark experiment on heat transfer in a model subassembly of a liquid metal cooled reactor was exercised as a standard problem in a working group meeting on the hydrodynamics and heat transfer organized and sponsored by the International Association of Hydraulics Research. The standard problem aims to analyze thermal and hydraulic characteristics in the model pin bundle under nonuniform geometrical and thermal conditions in square array of pins. A spacer grid exists in the pin bundle with variable heat production zones. The problem also intends to estimate the reliability and accuracy of codes used for thermal hydraulic analysis.

2. Problem Description

The model assembly of the BREST-type reactor core is a pin bundle of square arrangement as shown in Fig. 1. This bundle houses two zones differing in the pin diameter and heat generation. The model pin bundle contains one spacer grid. A eutectic alloy sodium-potassium (22% Na + 78% K) is used to cool the heated rods. The pin simulators are placed into the square wrapper which is mounted in the cylindrical vessel. The pin simulators are spaced by the bottom and top centering grids and by a transverse spacer grid [1].

3. CFD Analysis

A finite volume based code FLUENT® is used for analysis.

3.1 Geometry for FLUENT®

Figure 1 illustrates the geometry for FLUENT®. The mixing vane is not considered here so as to reduce the computational cost. Semi-fine mesh structures are used to model the flow with reasonable accuracy and speed. To accurately compute the heat transfer and viscous effect, mesh resolution in the region close to the wall becomes important. The size of the first mesh adjacent to the wall surface can be expressed as [2]

$$y_1^+ = \frac{y_1 u_\infty \sqrt{c_f/2}}{\nu} \tag{1}$$

Literature shows that size of the first mesh adjacent to the wall should be kept less than 40 to accurately calculate the secondary flow velocity and heat transfer. A reduction of 3 to 5 % in the heat transfer coefficient is reported for the case of a coarse mesh structure [3]. The first mesh size is kept between 45 and 50 in this study pursuant to Eq. (1).

The standard model is used for turbulence. This model predicts free shear flow spreading rates that are in close agreement with measurements for far wakes, mixing layers, and plane, round, and radial jets, and is thus applicable to wall-bounded flows and free shear flows [4].

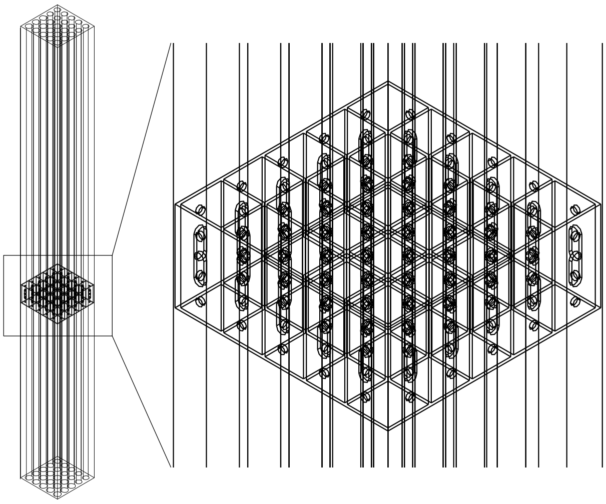

Figure 1. FLUENT® geometry.

3.2 Input Conditions

The tests had been performed in five different thermal conditions, varying the pin assembly power ration and inlet coolant temperature. The experimental study had indicated that, regardless of whether the power ratio was changed or not, the temperature rise was always higher on the side where fifteen pins with d_l= 14 mm were located. On the other hand, the code result showed that the higher coolant temperature rise occurred on the side where higher pin power was supplied [5]. In this study, the most incorrect case (condition 1) is selected for analysis which is summarized in Table 1.

Table 1. Thermal hydraulic input condition

Parameters	Condition 1
Inlet temperature (K)	328.99
Inlet velocity (m/s)	2.6
Density (kg/m^3)	865.023
Pin power$_{15}$ (kW)	1.35
Pin power$_{10}$ (kW)	2
Peclet number	1316
Reynolds number	53393

3.3. Calculational Results

Figure 2 shows the radial velocity distribution just underneath the grid. Note that the transverse convection is not dominant just underneath the grid.

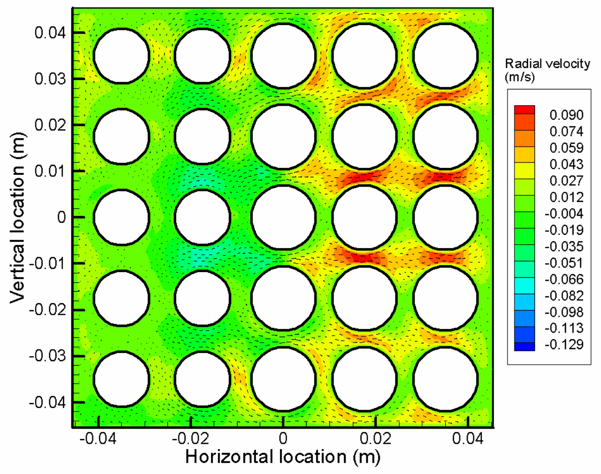

Figure 2. Radial velocity distribution at z = 382.5 mm.

Figure 3 illustrates the radial velocity distribution just above the grid. Observe that the momentum exchange has picked up, but not strong enough to be maintained along the coolant passage.

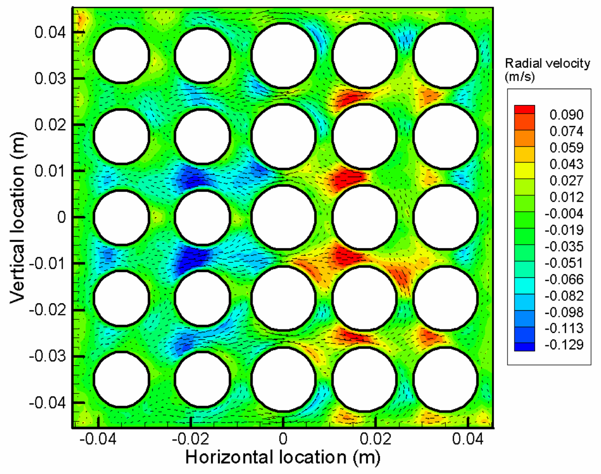

Figure 3. Radial velocity distribution at z = 455.5 mm.

The twisting effect is not generated, and only minor flow disorder is rendered. This mostly results from the "restoring of physics" of the coolant flow in the smooth pin simulators that is relatively too strong compared with the magnitude of flow disorder.

4. Conclusion

This computational study has shown that the currently imported geometry is not suitable for simulating the test results. Simplification of the mixing vane region has resulted in much weaker transverse flows than were observed from the experiment. In the experiment the effect of the spacer grid on the degree of "disorder" in the flow had persisted up to the bundle outlet, which was not reproduced in the computational analysis. More rigorous analytical work is underway to accurately model the mixing vane geometry and to compute the flow in the grid spacer region.

ACKNOWLEDGMENTS

This work was performed under the auspices of Center for Advanced Prototype Research Initiatives (CAPRI) as part of the Brain Korea 21 for the Department of Energy Systems Engineering funded by the Ministry of Education and Human Resources Development.

REFERENCES

[1] A.V. Zhukov, et al., Hydraulics and Heat Transfer in the Model Pin Bundles with Liquid Metal Coolant, 11th Meeting of the Int. Association for Hydraulic Research Working Group, pp. 138-172, 2004.

[2] W. Kays, M. Crawford, and B. Weigand, Convective Heat and Mass Transfer, McGraw-Hill, New York, NY, USA, pp. 187-188, 2005.

[3] X. Cheng and N.I. Tak, CFD Analysis of Thermal-hydraulic Behavior of Heavy Liquid Metals in Subchannels, Nuclear Engineering and Design, Vol. 236, No. 1, pp. 1874-1885, 2006.

[4] D.C. Wilcox, Turbulence Modeling for CFD, DCW Industries, Inc., La Canada, CA, USA, 1998.

[5] H.M. Son and K.Y. Suh, Solution to a Standard Problem Hydraulics and Heat Transfer in a Subassembly Model Cooled by Liquid Metal, Proceedings of the 5th Korea-Japan Symposium on Nuclear Thermal Hydraulics and Safety (NTHAS5), Jeju, Korea, November 2006.