Modeling of Flow Blockage for Simulation of Steam Convection and Reflooding of 5×5 Rod Bundle Experiments in ATHER

Young Seok Bang[a*], Byung Gil Huh[a], Deog Yeon Oh[a], Joosuk Lee[a], Jongrok Kim[b], Sang-Ki Moon[b]
[a]*Korea Institute of Nuclear Safety, 62 Kwahak-ro, Yuseong, Daejeon, Korea*
[b]*Korea Atomic Energy Research Institute, 111 Daedeok-daero 989beon-gil, Yuseong, Daejeon, Korea*
[*]*Corresponding author: k164bys@kins.re.kr*

1. Introduction

Cladding swell and rupture following a Loss-of-Coolant-Accident (LOCA) is recognized as a very important phenomenon in terms of its impact on Peak Cladding Temperature (PCT) as well as its starting point for Fuel Fragmentation, Relocation and Dispersal (FFRD) evaluation [1]. Accurately predicting when, where, and to what extent the swell and rupture occur has been a major concern in LOCA analysis [2].

In their previous study, the authors calculated a reflood experiment of Slab Core Test Facility (SCTF) simulating a 60% flow blockage using the MARS-KS code, which has been used as a regulatory auditing code [3]. In this work, hydrodynamic modeling of the flow paths in which blockage exist has shown the effect of flow blockage on thermal behavior of cladding at a reasonable level. However, for cases where more than 60 % flow blockages occur, significant uncertainty could exist in the prediction of reflood behavior. Also, it is emphasized that more reliable analysis is needed for interactions with ballooned fuel rods and cooling water.

In the Advanced Thermal-Hydraulic Evaluation of Reflood (ATHER) experiment program of Korea Atomic Energy Research Institute (KAERI), the effects of flow blockages have been studied for the various types of rod bundles. Among them, in the 5×5 rod bundle experiment, steam convection and reflood behavior under the condition simulated locally 90% blockage were observed [4]. Since these experiments are a detailed measure of the thermal-hydraulic behavior of individual rods and surrounding flow paths, calculating this behavior using a system code based on the lumped model probably constitutes a limitation of the coverage of the code. However, it can be a useful opportunity to assess how accurately current system codes predict the thermal behavior of ballooned fuel rods as long as it works.

A calculation of these experiments has been attempted using the 3D-vessel model (COBR-TF module) of the MARS code [5]. However, in the paper, it is not clear that the thermal behavior of ballooned rods, the prediction results from the 1D model, improvement of the accuracy by the 3D model, and its reasons have been sufficiently discussed.

In this study, these experiments are simulated using the MARS-KS code to evaluate the predictability of the code for thermal behavior of ballooned rods. The calculation starts with a one-dimensional single-channel model, investigating the necessary modeling for the implementation of ballooned rods, and applying it to multi-channel model and MULTID model.

Details of all experiment facility including the test data used in this study were obtained through available web searches and were used with the consent of KAERI. When the KINS-KAERI agreement for the use of the data is signed, analysis using formally released data will be carried out.

2. Experiments

Details of ATHER 5×5 rod bundle experiments are presented in the literature [4]. A 5×5 electrically heated bundle of rods, as shown in Fig. 1, features a sleeve inserted into a localized 3×3 rod array, simulating a 90% blockage of flow area. The diameter, pitch, and heating length of the rods are 9.5 mm, 12.85 mm, and 3.81 m, respectively, with all the rods producing uniform power and having axial power distributions of cosine shapes.

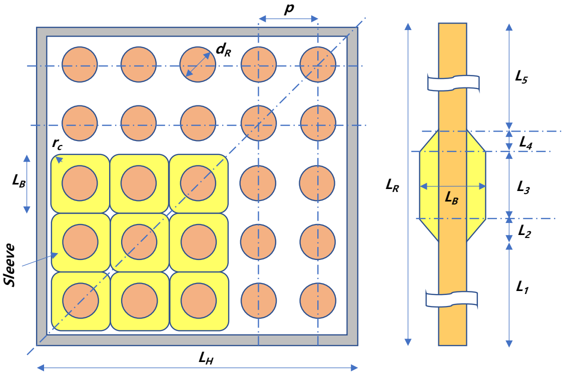

Fig. 1. Configuration of ATHER 5×5 rod bundle test

In this paper, a steam convection experiment and a reflood experiment were selected to evaluate the applicability of the MARS-KS code. The condition of the steam convection test is 18 kW for total heater power under 1 bar and 0.03 kg/sec steam flow rate. The condition of reflood experiment is 47 kW for total heater power under 2 bar in pressure, 90°C in subcooling of coolant temperature, and 2 cm/sec in flooding rate, respectively.

3. Code and Modeling

A code, MARS-KS-1.5 [6], the most recently released version, was used in this study.

3.1 Hydrodynamic Modeling

To simulate those experiments, an input model for hydraulic channels representing the test section and heat structures describing the heater rods was developed. As shown in Fig.2, three modeling cases were applied.

A. Single channel
B. Two channels (one for 9 ballooned rods and one for 16 intact rods) connected by crossflow junctions
C. MULTI-D component model in 2×2 meshes

In all cases, 29 nodes with non-uniform intervals in the axial direction were used, which allowed the location of the junction where the spacer grids were located to be close to the actual condition. Furthermore, a 5×5×29 MULTID component which describes all the flow paths of 25 rods individually has been attempted, and the minimum size of the node was 1 cm, making computational time steps as small as 10^{-7} sec under two-phase conditions, making realistic calculations difficult. For this reason, we apply a 2×2×29 MULTID component that enables more realistic computation.

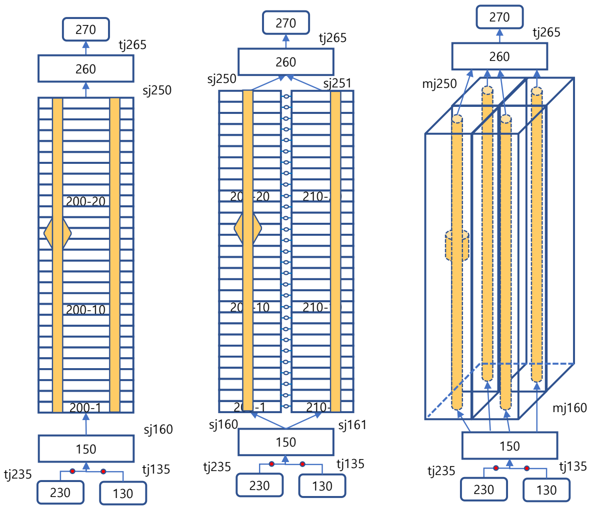

Fig. 2. MARS-KS modelings of ATHER 5×5 rod bundle test

To implement the sleeve for blockage inserted to 3×3 heater rods, flow volumes, junction areas, hydraulic diameters were reduced reflecting the sleeve location and their occupancy, respectively. In case of B and C, the hydraulic diameter at the location where blockages are present will have different values between channels, which in some cases can cause overheating of cladding due to severe friction losses. To avoid this, the hydraulic diameter of the overall mean of the section in an axial position is applied.

3.2 Thermal Modeling

In this study, the ballooned and un-ballooned sections of the heater rods were modeled as separate heat structures, and as a result, the middle parts of the nine heater rods were implemented as a heat structure having a layer corresponding to the sleeve. Sensitivity analysis confirms that it is necessary to consider thermal conduction over the sleeve to properly describe thermal balance in the present small scale experimental facility. Therefore, the axial conduction between the heat structure with sleeve and one without sleeve may not be considered during reflood period, but the effect is not expected to be significant. The outer diameter of the heat structure with sleeve, d_s, is determined to obtain the heat transfer area outside the sleeve.

In this process, it was necessary to introduce a fouling factor [6], a resistance to heat flow. As shown by the actual configuration of the test section, the part where the sleeve is in contact with the nearby sleeve does not contribute to the heat transfer to the fluid and should be reflected. The final value of the fouling factor will be determined by comparing the steady-state calculation results with the initial state of the experiment and detailed adjustments based on it. Figure 3 shows a concept of the thermal modeling.

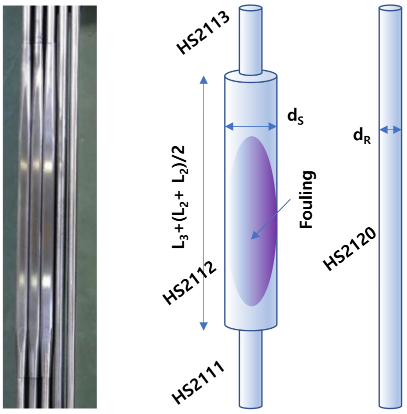

Fig. 3. Concept of thermal modeling of rod bundle

3.3 Boundary Condition

The calculations for the steam convection experiment were carried out in a steady state, with boundary conditions, pressure at the exit, injection flow rate and steam temperature at the inlet.

The reflood experiment was calculated through the steady state run to obtain the initial conditions and the subsequent transient run. Pressure at the outlet, injection flow rate, and the subcooling at the entrance were imposed, respectively.

The heat accumulated in the housing from the fluid portion and the heat loss through the housing can have a significant effect on thermal behavior. To take this into account, we modeled the housing as a heat structure. Insulation conditions at the outside of housing was imposed by accommodating the previous computational experience with similar experimental facility [7].

4. Results and Discussions

4.1 Steam Convection

Figure 4 shows the comparison between the results of the cladding temperature distribution calculated using the three previously defined modeling schemes and the experimental data. The calculation results are generally close to the experimental data, and in particular, local temperature increases in the ballooned portion are adequately predicted. This confirms that the current modeling scheme adopting the sleeve layer and the associated fouling factors was properly set. With the application of 2-channel modeling, the temperature increase in the ballooned portion is slightly over-estimated, and this effect continues to occur downstream. With MULTID modeling, three cladding temperature profiles can be obtained, covering the experimental data both upstream and downstream as well as in ballooned areas.

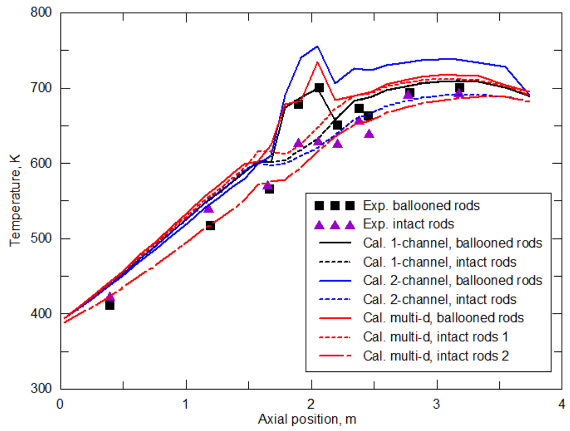

Fig. 4. Comparison of cladding temperature distribution

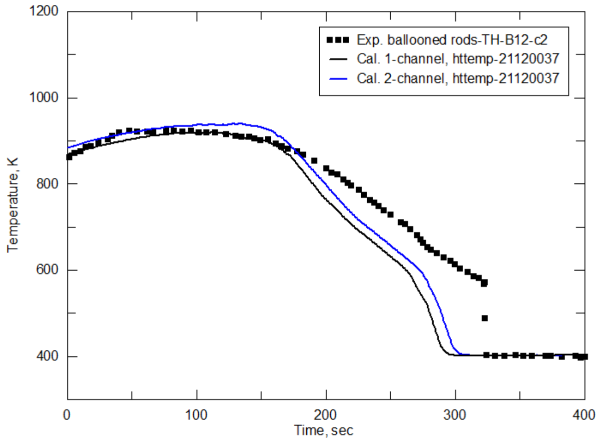

Fig. 5. Comparison of cladding temperature behavior

4.2 Reflood

Figure 5 shows a comparison of cladding temperatures at the ballooned location calculated by 1-channel model and 2-channel model with the experiment data. From this figure, we can see that the calculation results are well matched with the experimental data. With the 1-channel model, the final quenching was predicted a little faster than the experiment. With the 2-channel model, the peak cladding temperature is predicted to be slightly higher than the experimental data and the overall behavior is similar to the 1-channel case. Calculation using the MULTID component are in progress. Until now, the 1D-based calculation results are satisfactory enough, but it is hoped that more information can be obtained through the MULITD calculation.

5. Conclusions

An experiment of steam convection and one of reflood under the condition simulating locally 90% blockage at 5×5 rod bundle facility of ATHER were calculated using the MARS-KS code to evaluate the predictability of the code for thermal behavior of ballooned rods. In this paper, it was discussed how to model a heat structures that implements ballooned parts and a fouling factors that describe the contact between the rods. Based on the results so far, the modeling scheme discussed seems appropriate to predict the thermal behavior of ballooned rods during reflood period.

ACKNOWLEDGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety (KOFONS), granted financial resource from the (NSSC), Republic of Korea (No. 1805004-0118-SB110). KAERI's kind introduction to the experiment were very helpful. Authors would also appreciate Dr. B.D Chung for his helping us solve the code error.

REFERENCES

[1] USNRC, Fuel fragmentation, Relocation, and Dispersal during the Loss-of-Coolant Accident, NUREG-2121, 2012.

[2] Joosuk Lee et al, Evaluation of core-wide fuel pin burst during LOCA in APR1400, Nuclear Engineering and Design, Vol.371, 2021.

[3] Y.S. Bang et al, Multidimensional prediction of the effect of flow blockage in SCTF S1-01 experiment using MARS-KS code, Annals of Nuclear Energy, Vol. 152, 2021.

[4] KAERI, Safety Assessment and Model Improvement of Emergency Core Cooling, KAERI/RR-3945/2014.

[5] B. J. Kim et al, Reflood Phenomena in a 5x5 Ballooned Rod Bundle, Transactions of the Korean Nuclear Society Spring Meeting, Jeju, Korea, May 7-8, 2015.

[6] Bub Dong Chung, Model Improvement and User Requests of MARS-KS V1.5, NuSTEP Winter Meeting, 2018.

[7] Byung Gil Huh et al, Evaluation for KAERI 6×6 Reflood Test Using TRACE Code, Transactions of the Korean Nuclear Society Spring Meeting, Jeju, Korea, May 7-8, 2015.