CFD Verification of 5x5 Rod Bundle with Mixing Vane Spacer Grids

Park Sung-Kew [a*], Jang Hyung-Wook [a], Lim Jong-Seon [a], Park Eung-Jun [a], Nahm Kee-Yil [a]
[a] *1047, Daedeokdaero, Yuseong-gu, Daejeon, 305-353, Korea*
[*] *Corresponding author: skpark@knfc.co.kr*

1. Introduction

As a part of a new fuel assembly development, the critical heat flux (CHF) test is progressing on OMEGA CHF test facility located at CEA, Grenoble in France. CHF is the heat flux at which a boiling crisis occurs accompanied by a sudden decrease of the heat transfer at surface or deterioration of the heat transfer rate. Results of the CHF test are used for determining the CHF correlation, which is used to evaluate the thermal margin in the reactor core.

Computational fluid dynamics (CFD) has been used to save the time and cost for experimental tests, components design and complicated phenomena in all industries including the reactor coolant system. L. D. Smith et al. [1] applied the CFD methodology in a 5x5 rod bundle with the mixing vane spacer grid using the renormalization group (RNG) k-epsilon model. This CFD model agreed reasonably well with the test data. M. E. Conner et al. [2] conducted experiments to validate the CFD methodology for the single-phase flow conditions in PWR fuel assemblies. In this validation case, the CFD code predicted very similar flow field structures as the test data.

In this study, a CFD simulation under single-phase flow condition was conducted for one specific condition in a thermal mixing flow test of 5x5 rod bundle with some mixing vane spacer grids.

2. Thermal Mixing Flow Test

A thermal mixing flow test is carried out prior to water CHF test to quantify the mixing effect of test assembly. Thermocouples located at the center of each sub-channel to measure the coolant temperatures at the end of heating length. The data obtained by the mixing test are evaluated to determine the empirical spacer grid mixing factor, thermal diffusion coefficient (TDC). To evaluate this mixing factor (TDC), the measured exit sub-channel temperatures are compared with the predicted temperatures calculated by the sub-channel analysis code. In this study, measured data are also compared with results of a CFD simulation.

Figure 1 shows the structure domain of the test bundle. The test conditions are as follows;

- Pressure, *MPa* : 10.01
- Mass flow rate, *kg/s* : 2.617
- Inlet temperature, ^{o}C : 175.9
- Power, *MW* : 0.582
- Radial power distribution : Non-uniform
- Axial power distribution : uniform

The heater rod and sub-channel identifications of the test section are also shown in Figure 2.

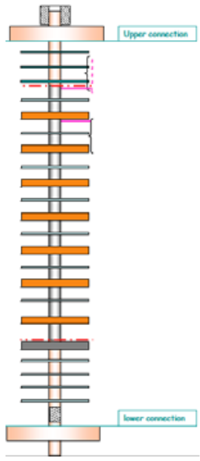

Figure 1. Structure domain of test bundle

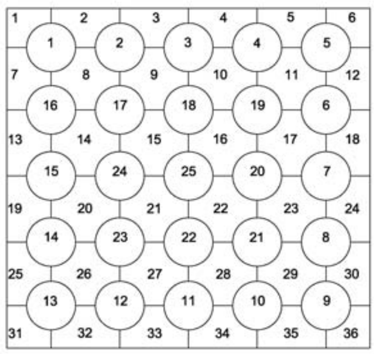

Figure 2. Test section configuration

3. Model and Result for CFD Analysis

The analysis for the thermal mixing flow test was performed using a CFD thermo-fluids analysis software, FLUENT 14.5 Version. The heating length of the CFD analysis domain is 3 m and the simple support grids located in the middle length were not considered because their effects are negligible. The mesh was generated by T-grid (fluent meshing) as shown in Figure 3. T-grid has powerful boundary repair and mesh generation tools, which can be used to repair and refine the surface meshes, generate surface wrapper mesh on highly complex and 'dirty' geometry, grow prism boundary layers on complex geometry, and advanced control of volume meshing. y+ and mesh sensitivity tests were only conducted on the smaller analysis domain with a mixing vane spacer grid because it was difficult to analyze axially the whole length of the flow field in the test bundle due to the limitation computer hardware resources. y+ means a non-dimensional wall distance for a wall-bounded flow. When dealing with turbulent flows like this test bundle, y+ with the most suitable near-wall treatment is very critical for a successful prediction of wall bounded turbulent flows. The calculated y+ with 3 prism layers is within 5. At this time, k-epsilon standard turbulence model was used. According to the Reference 3, this value satisfies the viscous sub-layer region. Figure 4 presents the result of mesh sensitivity test. From these results, the cell size options corresponding to approximately 21,000,000 of mesh number were applied to mesh the test bundle. The test bundle was divided into four parts: three parts with two mixing vane spacer grids and one part with one mixing vane spacer grid. Total number of mesh was about 90,000,000. Figure 5 shows the temperature distributions of the test, CFD and sub-channel analysis code for each sub-channel region at the outlet region of test bundle. Compared to the test result, the outlet temperature differences for CFD and sub-channel analysis code are within approximately 1.2 %. CFD analysis result is well suited to especially the sub-channel analysis code as well as test result. Figure 6 presents the maximum and average temperature distributions of CFD and sub-channel analysis code at axial length, respectively. As shown in result of CFD_MAX, it is identified that some part-peak points come out just after passing the mixing vane and the heat transfer is increased near the mixing vane and the influence of the mixing vane is kept in a short range. The result of CFD_MAX is the value of the maximum local point among all sub-channels. But, the result of CODE_MAX is the value of the maximum sub-channel among all sub-channels because the sub-channel analysis code is calculated for each sub-channel as a minimum cell. The average temperature distributions of all sub-channels in each axial location analyzed by CFD and CODE are almost same. Figure 7 shows temperature distributions around the hotter rod surfaces (#18, 25). Although the radial power of rod #25 is highest among those of the test rods, the rod with the hottest surface temperature is #18 and its maximum surface temperature is 515.1 *K* at sub-channel #10 of 2,740 mm of axial length. Figure 8 shows the temperature distribution from the spacer grid including the sub-channel #10 surrounding the rod #18. The third point corresponding to approximately 2,660 mm of axial length is the location just passing the spacer grid mixing vane. The influence of the temperature decrease caused by the spacer grid mixing vane represents after 60 mm from the spacer grid mixing vane location.

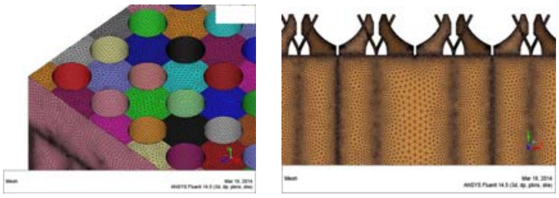

Figure 3. Mesh configuration

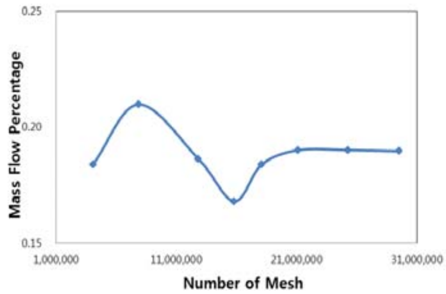

Figure 4. Mesh sensitivity test

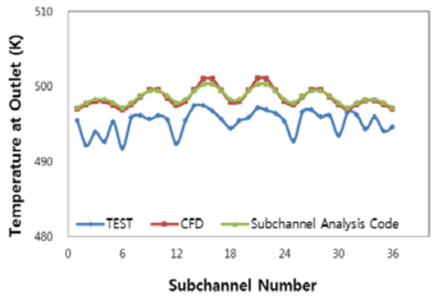

Figure 5. Comparison of temperature distribution at outlet region

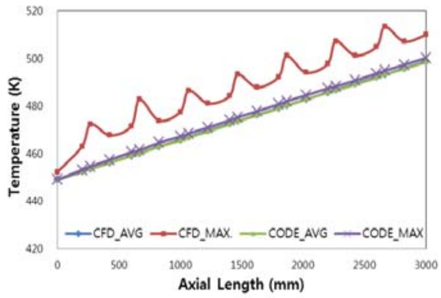

Figure 6. Maximum and average temperature distributions at axial length

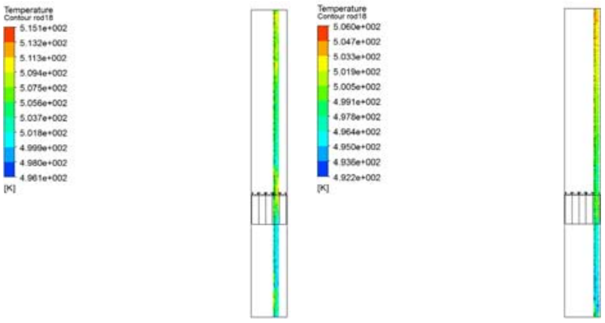

Figure 7. Temperature distributions around the hotter rod surfaces (rods #18, 25)

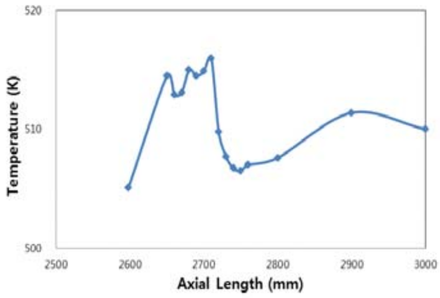

Figure 8. Temperature distribution from last spacer grid (sub-channel #10)

4. Conclusions

In this study, a CFD simulation under a single-phase flow condition was conducted for one specific condition in a thermal mixing flow test of 5x5 rod bundle with the mixing vane spacer grids to verify the applicability of the CFD model for predicting the outlet temperature distribution.

FLUENT 14.5 Version was used in this CFD analysis. For the successful prediction of the wall bounded turbulent flows, the y+ with 3 prism layers was determined within 5. At this time, k-epsilon standard turbulence model was used. The temperature distribution of CFD for each sub-channel at the outlet region of test bundle showed the difference approximately within 1.1% and 0.2% while comparing to that of test and sub-channel analysis code, respectively.

This single-phase CFD can play a significant role in developing fuel assembly and its components and performing the pressure drop and the heat balance tests for CHF test.

REFERENCES

[1] L. D. Smith, et al., “Benchmarking Computational Fluid Dynamics for application to PWR fuel,” 10th International Conference on Nuclear Engineering, 2002.

[2] M. E. Conner, et al., “CFD Methodology and Validation for Single-Phase Flow in PWR Fuel Assemblies,” Nuclear Engineering and Design, 240, pp. 2088-2095, 2009.

[3] S. M. Salim, et al., “Wall y+ Strategy for Dealing with Wall-bounded Turbulent Flows”, IMECS 2009 Vol II, 2009.