CFD Analysis for the Experiment of a Single Channel Post-Blowdown

Hyung Seok Kang, Bo Wook Rhee, Byung-Joo Min

KAERI, P.O Box 105, Yuseong, Taejon, Korea, 305-600, hskang3@kaeri.re.kr

1. INTRODUCTION

A CFD benchmark calculation for a post-blowdown test of the CANDU 28-element fuel bundle was performed to develop the CFD analysis methodology. This CFD analysis will be used to support the verification work of CATHENA code for the post-blowdown event. The CFD code can be effectively used on the analysis of the CANDU fuel channel where the thermal hydraulic behavior, especially for a radiation heat transfer between several heat structures, may be strongly dependent on the complicated geometry including the space grid in the fuel channel. The CS28-1 test was designed to understand the heatup and the swelling phenomena of the fuel element and the pressure tube in the post-blowdown event by AECL[1,2]. The CFX5.7 using the coupled solver algorithm was used for this calculation.

2. POST-BLOWDOWN TEST (CS28-1) [1]

The experimental facility consists of a test section of the 28-element fuel bundle (Fig.1) including the calandria tube, a cooling water tank and a boiler to produce a superheated steam. In this paper, we will only treat the steady state phase of the CS28-1 test where the chemical reaction on the surface of the FES(Fuel Element Simulator) didn't happen. A 10 kW power was supplied to the heater simulating the FES. The test section annulus had a gap between the PT(pressure tube) and CT(the calandria tube), through which CO_2 gas of 6 l/min flowed to maintain the oxide layer on the outside of the PT. The test was started by providing superheated steam of about 700 ^oC at 1 bar into the test section with 10 g/s. As for the results of the test, about 78% of the heat generation was transferred from the FES to the moderator tank by the radiation heat transfer.

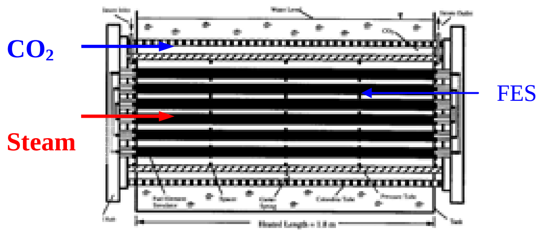

Figure 1. Side view of CS28-1 Test Section

3. CFD ANALYSIS

3.1 Grid Model and Boundary Conditions

A full grid model simulating the test section is generated like that in Fig. 2. The reason we developed the full model is to simulate a non-uniform steam temperature distribution of about 100 ^oC which happens due to the condensation at inlet region in the CS28-1. The cooling water tank, its bulk temperature is 40 ^oC, is treated as the boundary condition on the outside surface of the CT. The number of the mesh in the grid model is 4,324,340 cells. A 180 mesh cells in the z-axial direction are distributed to escape the high aspect ration along the length of 1.8m. The emissivity value on the FES surface, the inside and the outside surface of the pressure tube is 0.8, which is just quoted from the input of CATHENA[2]. However, the emissivity value on the inside surface of the CT in is 0.57 even though that of the metal is generally about 0.3~0.4 and that of the CATHENA input is 0.34[2]. Because we expect that the oxide layer at annulus gab can increase it, and the amount of the radiation heat transfer rate in the CFD calculation using 0.57 shows a valid value based on the several sensitivity calculations.

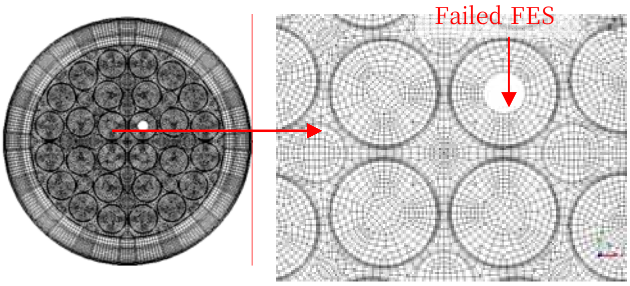

Figure 2. Grid Model in the CFD Calculation

3.2 Flow Field Model and Heat Transfer Model

The fluid flow and the heat transfer phenomena in the high temperature fuel channel are treated as a compressible flow, a highly turbulent flow, a conduction, a convection and a radiation heat transfer. The governing equations used in this calculation are the Navier-Stokes and the energy equation with the coupled solver algorithm. The discrete transfer model is used for the radiation heat transfer calculation. The generated mesh is too large to calculate on a single computer so that we use a parallel computing system.

3.3 Discussion on the CFX Results

The result of the heat balance calculations and the temperature of the inner, the middle and the outer FES during a steady state in the CFD calculation are shown in Table 1 and Fig. 3. And also, the temperature of the PT and the steam with the temperature measurement locations in the test section and the CFD calculation are shown in Fig. 3. We can see that most of the heat source given by the user input, about 81.7 %, is transferred into the cooling tank from the FES by the radiation heat transfer. This value is very similar to that of the test result. And also, this fraction is mainly dependent on the emissivity value on the inside surface of the CT. It can be explained by the role of the ε_2 term in the equation (1). If only the value of ε_2 is changed from 0.37 to 0.54, the amount of the heat flux from the PT to CT is increased by two times. This amount also affects the fraction of the convection heat transfer by the steam, and it determines the range of the steam temperature distribution along the FES in the CFD calculation. And also, the calculated steam temperature gives a great effect on the FES temperature distribution. According to the comparison the steam temperature at some locations in the test results(TC63~TC67) with those of the CFD results, the CFD results show higher temperature at the center region and lower temperature at other regions than those of test data within the range of 10 ^{o}C ~ 20^{o}C.

Table 1. Heat Balance Calculation in Steady State

Heat Source (FES)	Convection H. T (Steam)	Radiation H. T (Calandria Tube)
9.842 kW	1.486 kW	8.041 kW

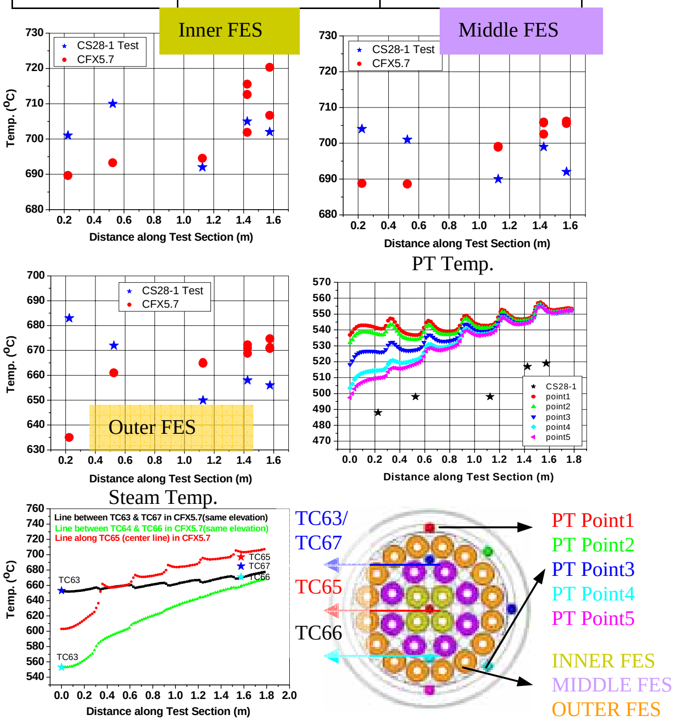

Figure 3. Comparison of Test Data and CFD Results

$$q_{12} = \frac{\sigma A\left(T_1^4 - T_2^4\right)}{\frac{1}{\varepsilon_1} + \frac{1-\varepsilon_2}{\varepsilon_2}\left(\frac{r_1}{r_2}\right)} \qquad (1)$$

According to the temperature distribution of the inner, the middle and the outer FES in the Fig. 3, the test data show almost constant values or a slightly decreasing trend along the test section. This is a very interesting phenomenon, when considering that the steam temperature in the test data is increased along the test section by 30^{o}C~100^{o}C. However, the FES temperature distributions of the CFD results show a slightly over estimated value by 10^{o}C~20^{o}C. And also, the predicted temperature distributions of the PT are higher than those of the test data by 30 ^{o}C ~50^{o}C. From the comparison work, we can see that the CFD calculation with the emissivity value of 0.57 on the inside of the CT shows a good result as a whole even though some discrepancy exists. We used the constant value along the axial direction.

4. CONCLUSION AND FURTHER STUDY

The results of the CFD benchmark calculation for the test of the post-blowdown event show a good agreement with those of the test data as a whole. However, to predict better the test data, a proper selection on the emissivity value of the inside surface of the CT considering the temperature variation of it is needed because it can govern the whole heat transfer mechanism and the temperature distribution in the fuel channel.

ACKNOWLEDGMENTS

This work was financially supported for the nuclear R&D program from the Ministry of Science and Technology of Korea. The authors are sincerely grateful for the financial support.

REFERENCE

[1] Q. M. Lei, D. B. Sanderson, K. A. Haugen and H. E. Rosinger, Post-Test Analysis of the 28-Element High-Temperature Thermal-Chemical Experiment CS28-1, *Proceedings of International Conference on Simulation Methods in Nuclear Engineering*, Montreal, Canada, 1993.

[2] H.S. Lim, K.M. Lee, and et al., “Validation of CATHENA against the CS28-1 High Temperature Experiment Under an Impaired Cooling Condition”, *Ann. Nucl. Energy*, **Vol.22**, *No.9*, pp. 593-600, 1995.

[3] H. S. Kang, B. W. Rhee and B-J Min, “CFD Analysis against the steady state experiment for Post-blowdown Phenomena in the CANDU fuel channel”, *Proceeding of the ’04 KNS Autumn Conf.*, Yongpyong, Korea, 2004.