CFD Analysis for the Steady State of a Post-Blowdown Experiment

Hyung Seok Kang, Bo Wook Rhee, Joo Hwan Park

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

1. INTRODUCTION

A CFD (Computational Fluid Dynamics) benchmark calculation for the steady state phase of a post-blowdown experiment (CS28-1) in a single high fuel channel [1] was performed to assist the development of the an accident analysis program for a CANDU-6. This CFD analysis was designed to support the verification work of the CATHENA code for the post-blowdown event, because the post-blowdown phenomenon was dependent on the complicated geometry of the fuel, especially for a combined radiation and convective heat transfer between the heat structures and the steam/CO_2 flow. And also the amount of thermal radiation absorption by steam and CO_2 may affect the temperature distribution of the fuel channel. The CFX5.7 using the coupled solver algorithm was used for the present calculation.

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

The experimental facility consisted of a test section of a 28-element fuel bundle (Fig.1) including the calandria tube, a cooling water tank and a boiler to produce a superheated steam. 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 the CT (Calandria Tube), through which CO_2 gas at 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 ℃ at 1 bar into the test section with 10 g/s. As for the results of the test, about 7.8±1.3 kW of the heat generation was transferred from the FES to the moderator tank by a radiation heat transfer.

3. CFD ANALYSIS [2]

3.1 Grid Model and Boundary Conditions

A full grid model of the FES to the CT simulating the test section (Fig. 1) was generated, because a non-uniform steam temperature of about a 100 ℃ difference at the inlet region may have a large effect on the heat transfer phenomenon. The cooling water tank with its bulk temperature of about 40 ^{o}C, was treated as a boundary condition on the outside surface of the CT. The number of meshes in the grid model was 4,324,340 cells including 180 cells along the axial direction. As for the boundary condition, a heat source condition simulating the electric heater power of 10 kW was given according to the power ratio [1]. The assumed steam temperature distribution at the inlet region was as shown in Figure 2. The pressure outlet boundary condition was set at the outlet region for the steam and the CO_2 in the test section. The temperature dependent properties of the heat structures of zirconium, alumina and graphite in the test section were used for the CFD input [2]. The emissivity value on the FES surface, the inside and the outside surface of the pressure tube, and the inside surface of the CT were 0.8, which was quoted from the input of CATHENA [1]. And also, the emissivity value of the space plate was assumed as 0.8 because the material of the space plate was the same as that of the FES. Plank-mean absorption coefficient [3] was used for a radiative heat transfer of steam and CO_2 in the fuel channel.

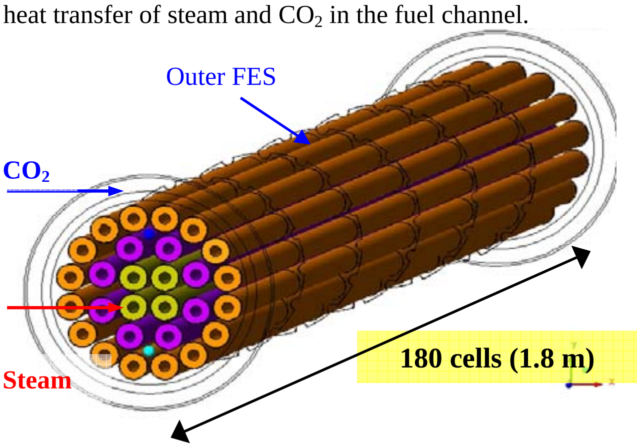

Figure 1. Grid Model in the CFD Calculation

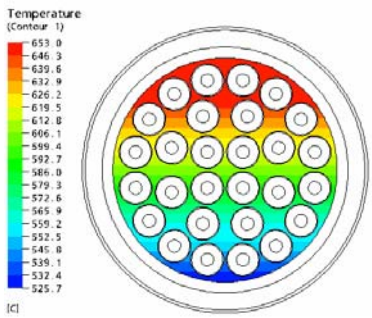

Figure 2. Assumed Steam Inlet Temp. Distribution

3.2 Flow Field Models and Heat Transfer Models

The fluid flow and the heat transfer phenomena in the high temperature fuel channel were 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 were the Navier-Stokes and the total energy equation with a coupled solver algorithm. The discrete transfer method [3] was used for the radiation heat transfer calculation.

3.3 Discussion on the CFX Results

The result of the heat balance calculations and the temperature of the steam and CO_2 including the temperature measurement locations in the CFD calculation were shown in Table 1 and Fig. 3. Most of the heat source given by the user input, about 81.9 %, was transferred into the cooling tank from the FES by a radiation heat transfer. The steam temperature (Fig. 3) at some locations in the CFD results (TC63~TC67) compared with those of the test showed higher temperature of about 3% at the center region (TC67) and a lower temperature of about 2% at the upper region and almost the same temperature at the bottom regions. The higher temperature at TC67 in the CFD results may be caused by the steam absorbed thermal photons and a non-mixing with the other steam of a lower temperature when flowing into the center hole of the space plate. However, this difference was small when considering the uncertainty of the test. The comparisons result of the pressure tube showed that the temperature difference of about 30~50 ℃ at the outlet region was large when compared with the steam and FES temperature [2]. It may be caused by the fact that the CO_2 enthalpy increase due to the absorbing thermal photons was overestimated.

Table 1. Heat Balance Calculation at Steady State

Heat Source (FES)	Convection H. T (Steam / CO_2)	Radiation H. T (CT outer surface)
9,841 W	1,660 / 57.8 W	8,061 W
Thermal Energy Increase (Absorption – Emission)		
Steam	145.2 W	(5633.2 - 5488.0) W
CO_2	2081.5 W	(10648.2 - 8566.7) W

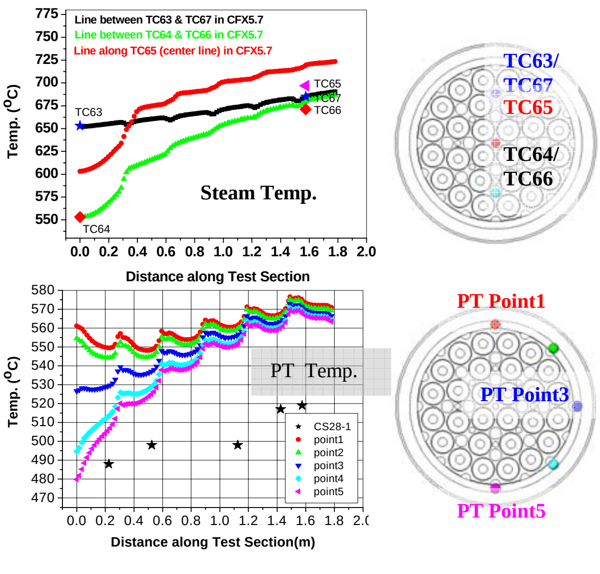

Figure 3. Comparison of Test Data and CFD Results

The emission (κI_b) and absorption (κI) of thermal photons by CO_2 during a radiation heat transfer was calculated by Eq. (1) [3] where I and I_b represented the local and blackbody intensity, and κ meant the absorption coefficient, respectively.

$$\frac{dI}{ds} = \kappa\left(I_b - I\right) \qquad (1)$$

In the CFD calculation, the estimated difference between the absorption and emission was 2081.5W. It may be relatively large, when considering the very narrow length of the annulus gap. And also, Plank-mean absorption coefficient used in the CFD calculation may have some errors [4]. Therefore, the amount of radiation heat transfer through CO_2 should be recalculated after selecting the proper absorption coefficient.

4. CONCLUSION AND FURTHER STUDY

The CFD benchmark calculation for the post-blowdown test in a CANDU fuel channel was performed to develop the CFD analysis methodology which can be used in the safety analysis of a CANDU. The CFD results showed a good agreement for the trend of the test results as a whole. However, the CFD results overestimated the temperature of the inner/middle/outer FES at the entrance region [2] and the pressure tube temperature at the outlet region. To resolve these problems, the proper gas absorption coefficient of CO_2 should be found and sensitivity CFD calculation is necessary.

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 ICSM in Nuclear Engineering*, Montreal, Canada, 1993.

[2] H. S. Kang, B. W. Rhee and J. H. Park, “CFD Analysis for the Steady State Test Simulating High Temperature Chemical Reaction in the CANDU Fuel Channel (to be published)”, Technical Report, KAERI /TR-3114/2005, KAERI, 2006.

[3] Michael F. Modest, *Radiative Heat Transfer*, **2nd ed.**, McGraw-Hill, New York, 1993.

[4] O. J. Kim and T. H. Song, Data base of WSGGM-based spectral model for radiation properties of combustion products”, J. of Quantitative Spectroscopy & Radiative Transfer, 64, pp.379-394, 2000.