Transactions of the Korean Nuclear Society Autumn Meeting
Jeju, Korea, October 21-22, 2010

Three Dimensional Analysis of CANDU6 Fuel Channel Using CFX Code

Hyoung Tae Kim*, Wang Kee In, Joo Hwan Park
Korea Atomic Energy Research Institute, 1045 Daedoek-daero, Yuseong-gu, Daejeon 305-353, Korea
**Corresponding author: kht@kaeri.re.kr*

1. Introduction

This paper presents the numerical investigation of coolant flow in the CANDU6 fuel channel using the computational fluid dynamics (CFD) approach. We analyze the thermal-hydraulic behavior of coolant flow along bare fuel bundles with appendages of end support plate, spacer pad, and bearing pad. The computer code used is a commercial CFD code, CFX-12 [1].

The purpose of this work is to establish a detailed three-dimensional fluid flow and heat transfer model for a CANDU fuel channel loaded with a standard 37-element fuel bundle under the normal operating condition.

2. CANDU6 Fuel Channel Model

2.1 Grid Generation

Figure 1 shows the configuration of CANDU6 fuel elements with appendages.

The arrangement of fuel elements in the concentric rings of the CANDU6 fuel bundle results in different flow rates at the vacant gaps between the different rings of elements (i.e., coolant subchannels). This geometric configuration was simulated in the CFX analysis of CS28-2 experiment [2]. However, the complex flow in the subchannels of CANDU6 is highly turbulent, single or multi-phase and influenced by appendages. On basis of these phenomena, a hybrid mesh for the complex geometry such as spacer pad, bearing pad, and end support plate is generated for the O6 channel, which is a representative fuel channel among 380 channels in the CANDU reactor.

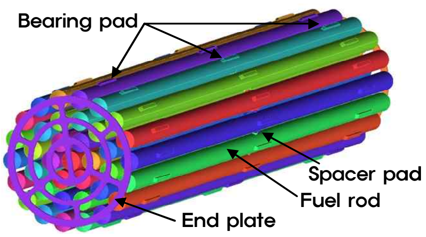

Fig. 1. Configuration of CANDU6 fuel bundle.

For the grid generation of CANDU6 fuel channel, the 2 dimensional surface mesh on the cross section of the fuel bundle is extruded to construct the 3 dimensional grid elements, and the grid elements for the appendage parts are replaced by the hybrid meshes generated independently. Figure 2 shows the grid generated for the fuel bundle including the hybrid meshes. The limited computer power for the large mesh memory size forced us to analyze only a 1/6 segment of the fuel channel. The number of the node for the present grid is 610,447.

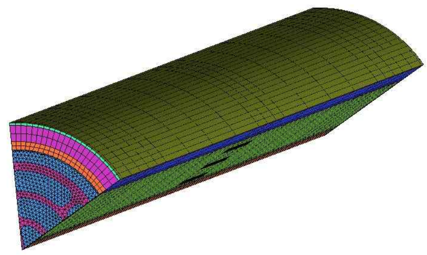

Fig. 2. Grid generation for the CANDU6 fuel bundle.

2.2 Boundary Condition

The thermal-hydraulic analysis of CANDU6 fuel channel has been performed by the one dimensional safety analysis code, CATHENA [3]. In this code model the bundle average power is applied to each bundle. However, in the present CFX model, the axial power profile along the local position of each bundle is assumed based on the variation of the average bundle power in different bundles. Finally the axial profile is obtained by the 3rd order polynomial as shown in Fig. 3.

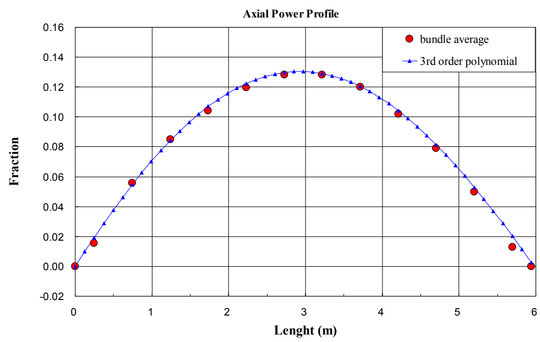

Fig. 3 Axial power profile along the length of the channel.

The boundary conditions are provided by the single channel analysis results by CATHENA code as shown in Table 1.

Table 1: Main Parameters used for Core Modeling

Parameter		O6 channel (full geometry)	1/6 model
Channel thermal power		7.3 MW	1.217 MW
Inlet coolant (11.0 MPa)	flow rate	24.34 kg/sec	4.057 kg/sec
	temperature	267.1℃	
	density	862.81 kg/m^3	
Outer surface Temp.		73℃	

3. CFX Anlaysis Results

3.1 Comparison with the CATHENA Results

Figure 4 shows the axial temperature distributions of the center fuel element (radial position between the fuel pellet and the gap) by the CATHENA and CFX-12 codes. We can see that CFX-12 predictions are in good agreement with those by the CATHENA code for the single liquid convection region (before axial position of the half of the channel length). Therefore, the CFX results for the 4th bundle are credited and investigated.

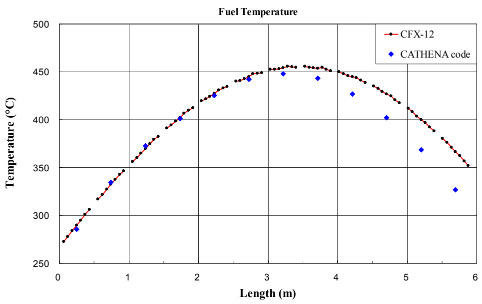

Fig. 4. Comparison of fuel temperature between the results of the CATHENA and CFX-12 codes.

3.2 Coolant Velocity

Figure 5 shows the variation of the velocity field at the local position of 269 mm from the inlet of each bundle. We can see that the variation of the velocity field at downstream of the 4th bundle is negligible. Furthermore, the 2-d velocity profile shows the cyclic pattern in each bundle, as coolant passes different geometric configuration of the internal structures of the fuel bundle.

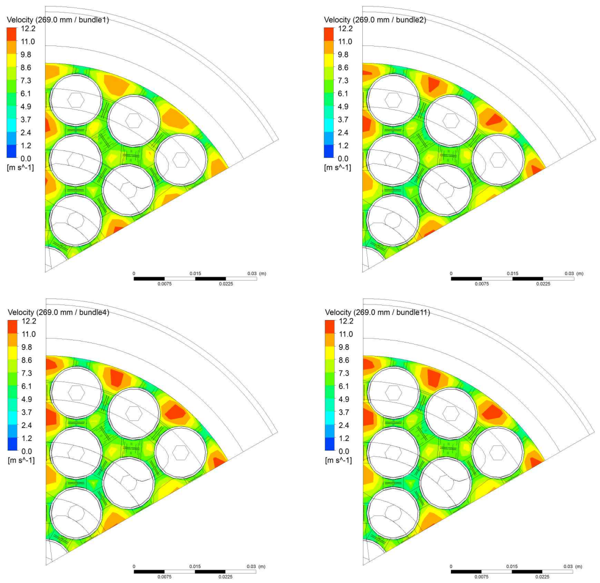

Fig. 5. Variation of coolant velocity along the axial bundle location.

3.3 Coolant Temperature

Figure 6 shows the variation of the temperature field at the local position of 269 mm from the inlet of each bundle. We can see that the temperature gradient becomes clear as the bundle power is increased along the axial channel length. The location of the hot spot on each temperature field is affected by the fuel rod appendages.

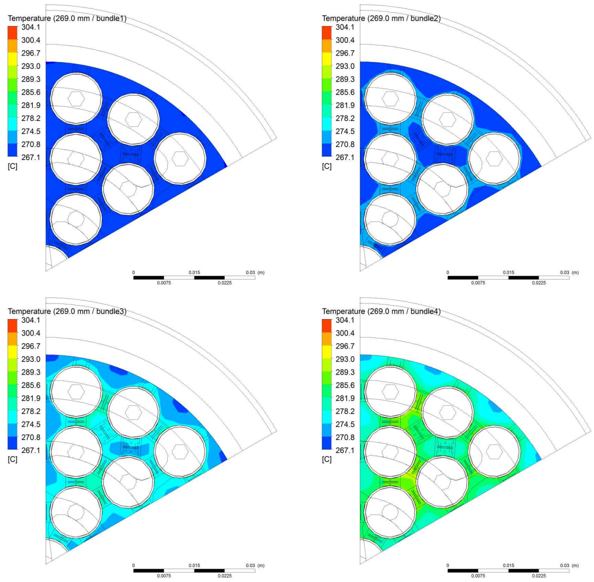

Fig. 6. Variation of coolant temperature along the axial bundle location.

4. Conclusions

The thermal-hydraulic behavior of coolant flow along bare fuel bundles with appendages of end support plate, spacer pad, and bearing pad is analyzed by the CFX-12 code. It is shown that the CFX-12 prediction of the fuel temperature is in good agreement with the CATHENA code result in the single liquid convection region.

The coolant velocity profile has the cyclic pattern in the fuel bundle with the effect of the internal structure of the bare fuel rods and their appendages. The change of velocity profile at the local position of the 4th bundle is negligible as coolant flowing through the downstream of this bundle location. Therefore, the CFX analysis with fine grid density can be considered for the 4th fuel bundle in the future work.

REFERENCES

[1] ANSYS CFX-Solver Modeling Guide, Release 12.0, ANSYS, Inc., April 2009.

[2] H.T. Kim, B.W. Rhee and J.H. Park, "CFX simulation of high temperature thermal–chemical experiment: CS28-2," Annals of Nuclear Energy, Vol. 35, No. 4, pp. 677~689, 2008.

[3] B.N. Hanna, "CATHENA: A Thermalhydraulic Code for CANDU Analysis", Nuclear Engineering and Design (180), pp. 113-131, 1998.