NURETH14-534

CFD ANALYSIS OF THE 37-ELEMENT FUEL CHANNEL FOR CANDU6 REACTOR

Hyoung Tae Kim, Bo Wook Rhee and Joo Hwan Park
Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yuseong-gu, Daejeon 305-353, Korea

Abstract

We analyzed the thermal-hydraulic behavior of coolant flow along fuel bundles with appendages of end support plate, spacer pad, and bearing pad, which are the CANDU6 characteristic design. The computer code used is a commercial CFD code, CFX-12 [1]. The present CFD analysis model calculates the conjugate heat transfer between the fuel and coolant. Using the same volumetric heat source as the O6 channel, the CFD predictions of the axial temperature distributions of the fuel element are compared with those by the CATHENA [2] (one-dimensional safety analysis code for CANDU6 reactor). It is shown that CFX-12 predictions are in good agreement with those by the CATHENA code for the single liquid convection region (especially before the axial position of the first half of the channel length). However, the CFD analysis at the second half of the fuel channel, where the two-phase flow is expected to occur, over-predicts the fuel temperature, since the wall boiling model is not considered in the present CFD model.

Introduction

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.

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 sub-channels). In the previous study [3, 4] this geometric configuration was simulated in the CFX analysis of the 28-element bundle under single-phase (steam) convection (CS28-2 experiment). However, the complex flow in the sub-channels 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 limiting fuel channel, called O6, which is a representative hot fuel channel among 380 channels in the CANDU6 reactor.

The present CFD analysis model calculates the conjugate heat transfer between the fuel and coolant. Using the same volumetric heat source as the O6 channel (representative channel for conservative analysis), the CFD predictions of the axial temperature distributions of the fuel element are compared with those by the CATHENA (one-dimensional safety analysis code for CANDU6 reactor).

The present CFD analysis accounts for comprehensive simulations of a full-scale CANDU6 fuel channel, for the first step, and will be used to predict the cross flows and their flow mixing and asymmetry, which cannot be captured by the one-dimensional safety analysis code.

1. CANDU6 Fuel Channel Model

1.1 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. 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.

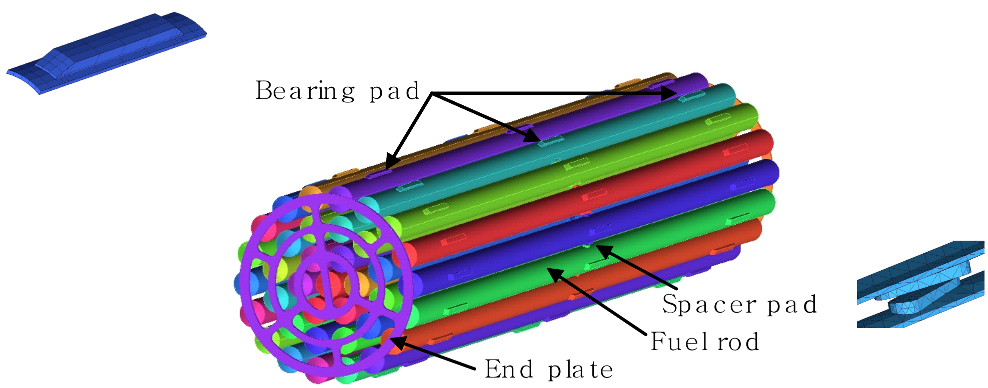
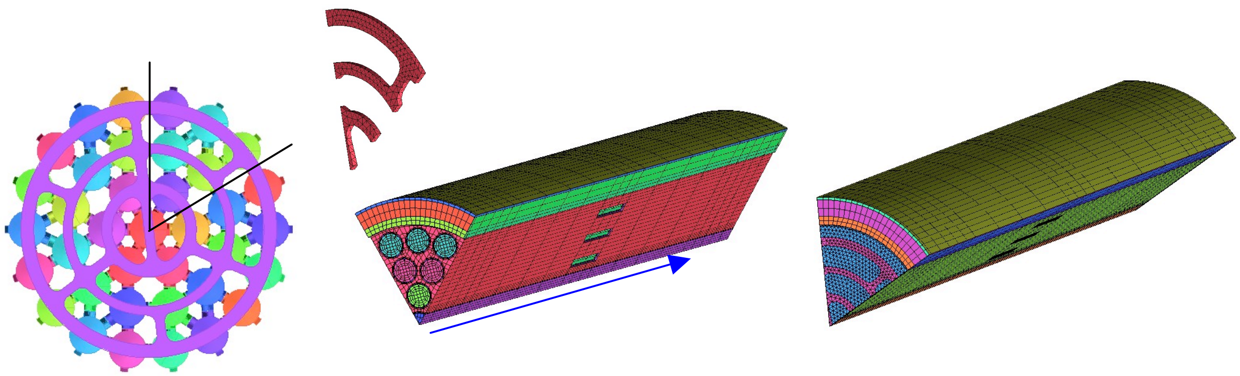


Figure 1 Configuration of CANDU6 fuel bundle.

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 structure of the fuel bundle, and the box of grid elements including the appendage parts are removed from this grid structure of the bundle. Then the hybrid meshes for the appendage parts which are generated independently are attached to the grid structure of the bundle. Figure 2 shows the grid generated for the fuel bundle including the hybrid meshes. The limited computer power for the large mesh memory size enforced us to analyze only 1/6 segment of the fuel channel. The grid generation software ICEM CFD [5] is used for this purpose. The number of node for the present grid is 610,447.



(a) End plate (1/6) (b) Extrusion of the 2-d mesh (c) Combination of the hybrid meshes

Figure 2 Grid generation for the CANDU6 fuel bundle.

1.2 Boundary condition

The thermal-hydraulic analysis of CANDU6 fuel channel has been performed by the one dimensional safety analysis code, CATHENA [2]. In this code model the bundle average power obtained from Reference [6] 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 4th order polynomial in the expression (1) and plotted in Fig. 3.

$$0.08011x - 0.00727x^2 - 0.00228x^3 + 0.00021x^4 \tag{1}$$

where, x is the distance (m) from the inlet of the fuel bundle.

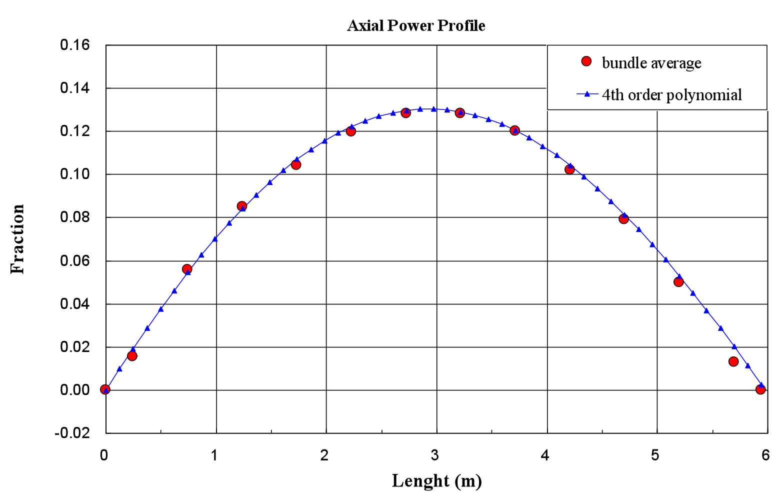


Figure 3 Axial power profile along the length of the channel.

The radial power distribution for the 4 circular rings consisting of number of elements is obtained from Reference [6] and given in Table 1.

Table 1 Radial power distribution

	Number of elements	Normalized radial power distribution
Center pin	1	0.021
Inner ring	6	0.1306
Middle ring	12	0.2986
Outer ring	18	0.5503

The boundary conditions are provided by the single channel analysis results by CATHENA code as shown in Table 2.

Table 2 Main parameters used for core modeling

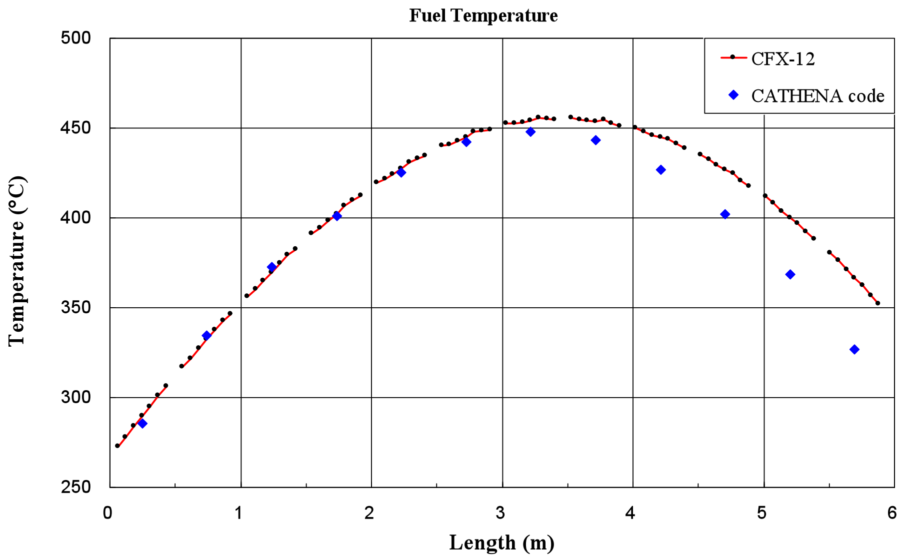
Parameter		O6 channel (full geometry)	1/6 model
Channel thermal power		7.3 MW	1.217 MW
Inlet coolant (11.0 MPa)	flow rate	24.34 kg/sec	4.057 kg/sec
	temperature	267.1℃	
	density	862.81 kg/m^3	
Pressure tube outer surface Temp.		73℃	

2. CFX Analysis Results

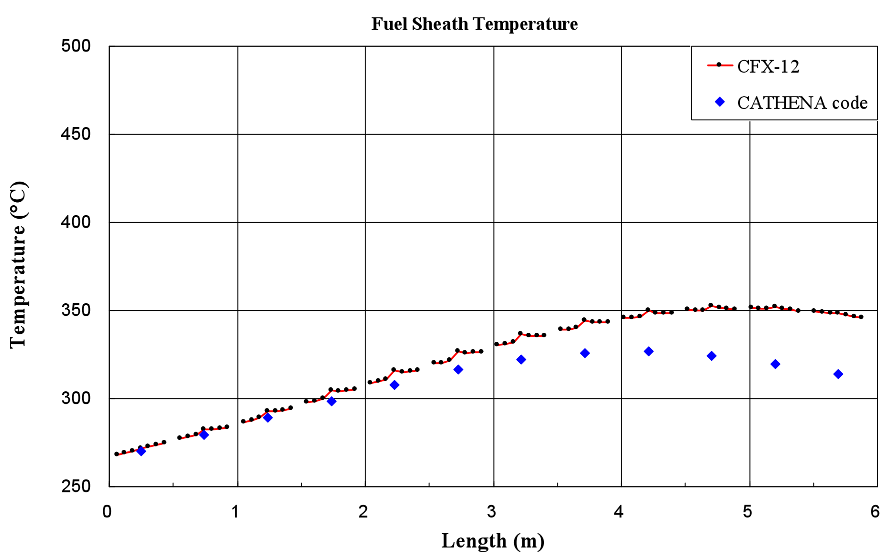
2.1 Comparison with the CATHENA results

Figure 4-(a) and 4-(b) show the axial temperatures of the center fuel element at the pellet outside and the sheath outside, respectively, 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). This single convection region is found up to the 4th bundle.



(a) Pellet outside temperatures



(b) Sheath outside temperatures

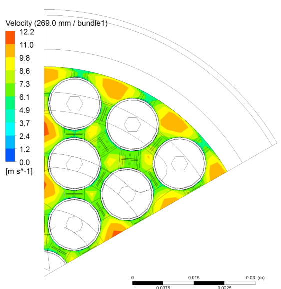
Figure 4 Comparison of fuel temperature between the results of the CATHENA and CFX codes.

2.2 Coolant velocity

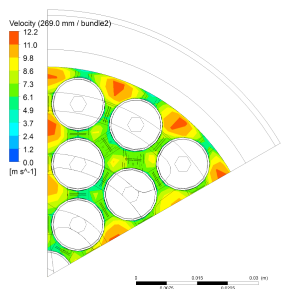
Figure 5 shows the variation of the velocity field at the local position of 269 mm from inlet of each bundle. We can see that the variation of the velocity field at downstream of 4th bundle is negligible. Furthermore, the 2-d velocity profile shows the cyclic pattern in each bundle, as coolant passing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

2.3 Coolant temperature

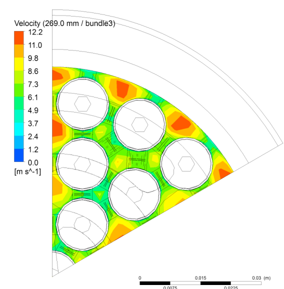
Figure 6 shows the variation of the temperature field at the local position of 269 mm from inlet of each bundle. We can see that the temperature gradient become clear as the bundle power is increased along the axial channel length. The location of the hot spot on each temperature field is affected by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



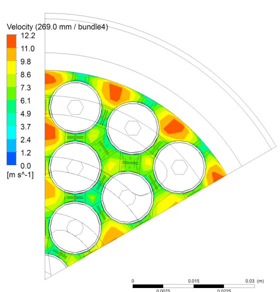
1st bundle



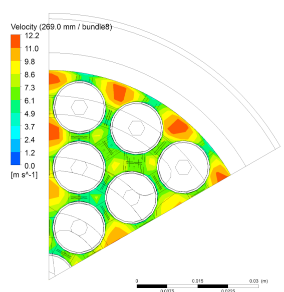
2nd bundle



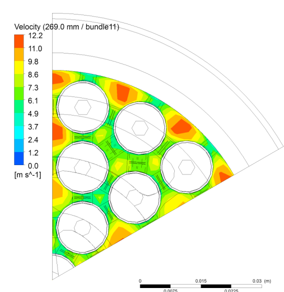
3rd bundle



4th bundle



8th bundle



11th bundle

Figure 5 Variation of coolant velocity along the axial bundle location (269 mm from inlet of each bundle).

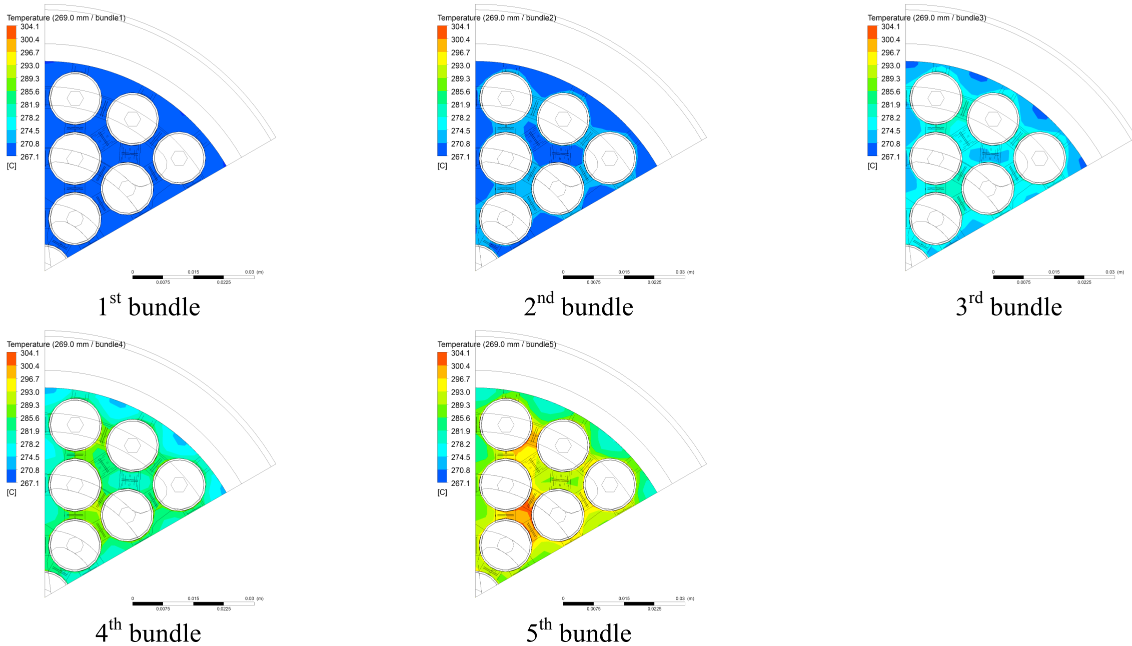


Figure 6 Variation of coolant temperature along the axial bundle location (269 mm from inlet of each bundle).

2.4 Pressure tube temperature

Figure 7 shows that the temperature near the hot solid part of the bundle (bearing pad) is higher than other region. The temperature difference between the maximum and minimum of pressure tube temperature is shown to be about 5 ^{o}C at the location of 269 mm from inlet of 4th bundle.

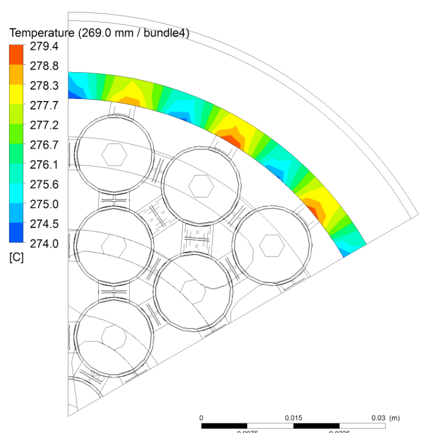


Figure 7 Temperature profile on the pressure tube (269 mm from inlet of 4th bundle).

3. Conclusion

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.

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.

It is shown that CFX-12 predictions are in good agreement with those by the CATHENA code for the single liquid convection region (especially before the axial position of the first half of the channel length). However, the CFD analysis at the second half of the fuel channel, where the two-phase flow is expected to occur, over-predicts the fuel temperature, since the wall boiling model is not considered in the present CFD model. It shows the instability and convergence problems, when the CFD analysis includes conjugate heat transfer and the wall boiling together. Therefore, the CFD wall boiling models with heat flux boundary condition on the fuel surface will be used to predict the fuel temperature and the coolant voiding at downstream of the CANDU6 fuel channel for the future work.

Acknowledgements

This work has been carried out under the Nuclear Research and Development program of Korea Ministry of Education, Science and Technology.

4. References

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

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

[3] H.T. Kim, B.W. Rhee and J.H. Park, “CFX simulation of a horizontal heater rods test”, CFD4NRS: OECD/NEA and IAEA Workshop, Munich, Germany, 2006 Sep. 5-7.

[4] 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, 2008, pp. 677-689.

[5] ANSYS ICEM CFD 12.0 User Manual, ANSYS Inc., 2009 April.

[6] B.J. Moon, “CATHENA Slave Channel for Wolsong Fuel and Fuel Channel Model”, 59RF-03500-AR-003, Rev. 1, 2009.