Transactions of the Korean Nuclear Society Spring Meeting
Gyeongju, Korea, May 29-30, 2008

Prediction of Heat Transfer in a CEA Extension Shaft Guide Tube

Seong Hoon Kim [a*], Young In Kim [a], Chun Tae Park [a]
[a] *Korea Atomic Energy Research Institute, 150 Deokjin-dong, Yuseong-gu, Daejeon 305-353, Korea*
[*] *Corresponding author: shkim822@kaeri.re.kr*

1. Introduction

The pressurizer of the SMART (System-Integrated Modular Advanced ReacTor) uses nitrogen gas as a pressurizing medium [1]. To minimize the portion of the saturated steam pressure in the system pressure, the temperature of the pressurizer is maintained at a lower temperature than that of the reactor coolant system. The pressurizer cooler cools down the pressurizer. The wet thermal insulator that covers the surface of the pressurizer reduces the heat transfer from the hot reactor coolant.

However, there are holes in the wet thermal insulator to provide spaces for moving the CEA (Control Element Assembly) Extension shafts. For these shafts the CEA Extension guide tubes are installed in the pressurizer. The coolant in the CEA Extension guide tube is connected to the reactor coolant, but it is completely isolated from the pressurizer water. Due to the temperature difference between the pressurizer water and the reactor coolant, natural convection occurs in this tube. The magnitude of the heat transfer by this natural convection is an important factor to decide the thermal size of the pressurizer cooler.

The magnitude of the heat transfer obtained by the computation fluid dynamics (CFD) is presented in this paper.

2. Methods and Results

2.1 Computational Methods

The computation was conducted by the commercial CFD code, FLUENT. This code solves the transport equation of the mass, momentum, energy, and turbulence quantities [2]. The gravity effect was included in the simulation because the main source of the heat transfer is the natural convection in the tube. It was also considered that the water properties, such as density, thermal conductivity, and viscosity, were varied with a temperature. The RNG k-ε model [3], which gives better predictions than the standard k-ε model for many cases, was selected as the turbulence model.

Figure 1 shows the schematic diagram of a computational domain. A single CEA extension shaft tube was chosen for the computation. A shorter tube was simulated to reduce the computational time, because it was expected that the heat transfer would mainly occurred in the lower part of the tube and the higher part of the tube would be relatively unimportant.

To provide correct boundary conditions to the computational model, a flow passage at the bottom of the pressurizer was added. The outflow boundary was placed at a long distance from the hole. The total number of grids is about a million. The inlet boundary condition was assigned to be a uniform flow of 0.5 m/s. The inlet flow temperature is 310^{o}C and the wall temperature of the CEA shaft guide tube is 100^{o}C.

A preliminary simulation reveals that the flowfield inside the tube shows a highly unsteady motion. The calculated heat flux during the iteration of a steady simulation showed a large magnitude of variation. After all, an unsteady simulation was conducted for a sufficient time to get a converged value of the estimated heat transfer rate. After a series of simulations, a time step was decided to be 0.1 sec.

2.2 Results

Figure 2 shows the estimated heat flux contours at the tube wall. The highest heat flux occurs near the lip of the tube. The reason of the high heat flux is that the hot flow along the shaft directly hits the cold tube wall and that the temperature difference between the wall and the flow is the highest at that position. In addition to the impingement of the hot flow on the wall, the natural convection increases the heat transfer in the tube. The fact that a simulation without the gravitation estimates a far smaller heat transfer rate supports this idea.

The estimated heat transfer rate is greater than expected. To reduce this heat transfer a simple idea was suggested; an orifice-like structure is installed at the tube bottom to reduce the opening.

A gap between the shaft and the orifice must be provided to move the guide tube. A simulation with a gap of 2 mm was conducted. The result shows reduced heat transfer rate (Table 1). The maximum heat transfer occurs at a little higher position from the tube bottom

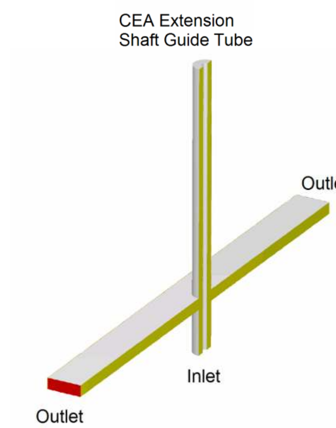
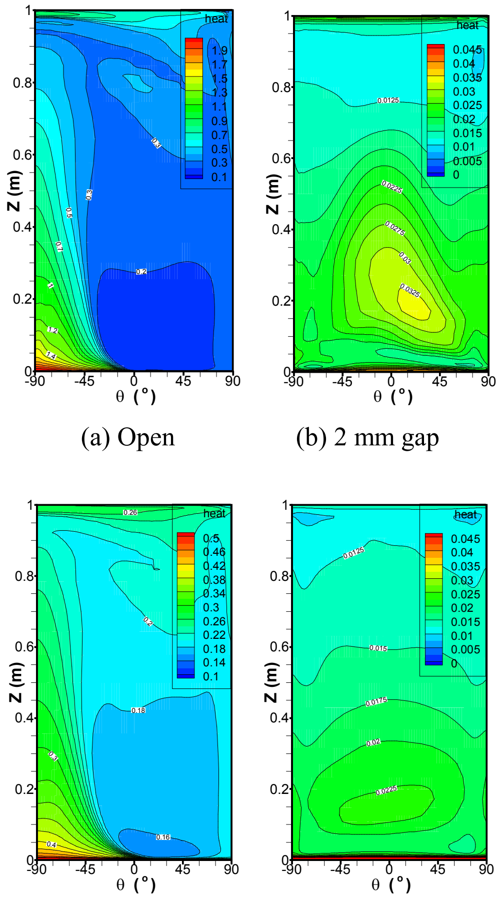


Fig. 1. Schematic diagram of the computational domain



(a) Open (b) 2 mm gap

(c) Open (d) 2 mm gap

Fig. 2. Heat flux contours (kW) at the CEA extension shaft guide tube wall (top: no wall conduction, bottom: wall conduction)

TABLE 1

Heat transfer rate (kW)

Case	Without wall	With wall
Open	123	75
2.0-mm-gap	6.0	6.2

(Figure 2-b). It means that the flow through the gap hits the wall at that position.

2.3 Wall Conduction

Unlike the pressurizer bottom, the guide tube is not covered with a wet thermal insulator. So, the wall conduction may have an effect on the heat transfer rate, especially at the tube bottom. Grids for the solid wall were added to computational domains, and the simulations were conducted.

Firstly, the simulation with the open hole estimates a reduced heat transfer rate when compared with the simulation without considering a wall conduction. When there is no wall, the temperature difference between the flow and the tube wall is high. However, the wall conduction reduces this temperature difference.

Secondly, when there is a 2-mm-gap at the tube bottom, the wall conduction case shows the similar estimated heat transfer rate to the case without considering the wall conduction. The wall conduction has little effect on the heat flux in this reduced opening case. This can be explained by the following. When there is a small gap, the temperature difference between the flow and the tube wall at the position of the highest heat transfer rate is reduced as the flow convects from the gap to that position. The wall existence may heat up the flow temperature, but this temperature rise is smaller than the temperature increment during the convection.

3. Conclusions

The heat flux through the CEA extension shaft guide tube wall was estimated by the CFD. We found the following:

1) The estimated heat flux is high due to the direct impingement of the hot reactor coolant on the cold tube wall.
2) An idea that the opening at the tube bottom must be reduced to block this direct impingement of the hot flow was suggested. This idea was checked by a simulation of the case in which an orifice-like structure reduces the opening at the tube bottom. The simulation showed a reduced heat transfer rate.
3) The simulation considering the wall conduction estimated a smaller heat transfer rate when there was no blockage at the tube bottom.

REFERENCES

[1] S. Q. Zee, et al., Development of Integral Reactor, KAERI/RR-2695, KAERI, Korea, 2005 (in Korean).
[2] FLUENT User's guide, FLUENT, 2003.
[3] D. C. Wilcox, Turbulence Modeling for CFD, 2nd ed., DCW Industries, Inc., California, USA, 1998.