CFD Validation of heat transfer in a spent fuel assembly

Seung Hun Yoo, Hyeun Min Kim, Hee Cheon NO
Korea Advanced Institute of Science and Technology,
373-1 Guseong-dong, Yuseong-gu, Daejeon, Republic of Korea,
yoosh@kaist.ac.kr, jeannette@kaist.edu, hcno@kaist.ac.kr

1. Introduction

The Computational Fluid Dynamics (CFD) is a sophisticated analysis technique used for predicting the hydraulics and heat transfer. In this paper, one of the optimized methodologies is proposed to apply for the analysis of heat transfer in a spent fuel assembly by using FLUENT 6. In order to develop the proposed methodology, the experimental data refers to a single assembly heat transfer test in Pacific Northwest Laboratory [1]. And the numerical calculation is compared with the experimental data to validate CFD analysis process.

Nomenclature

F_G view factor matrix on surfaces
h heat transfer coefficient [W/(m ^{2} °C)]
k thermal conductivity [W/(m °C)]
Gr Grashof number
Pr Prandtl number
Ra Rayleigh number
T temperature [K]
A heat transfer area [m ^{2}]
ρ helium gas density [kg/m ^{3}]
v specific volume of helium gas [m ^{3} /kg]
β helium thermal expansion coefficient [1/°C]
ε emissivity of solid wall surface
σ Stefan-Boltzmann constant [W/(m ^{2} °C ^{4})]

2. Single assembly heat transfer test

The single assembly heat transfer test performed at Pacific Northwest Laboratory using an electrically heated model of a PWR spent fuel assembly (Fig. 1). The fuel assembly geometry physically and thermally simulates a 15x15 rod matrix and a uniform axial heat generation of 1kW. The cask cavity is filled with helium and the pressure is maintained at positive pressure. A total of 98 separate thermocouples are mounted on the test section. And data collection for tests is performed at steady state conditions that are assured by monitoring thermal equilibrium.

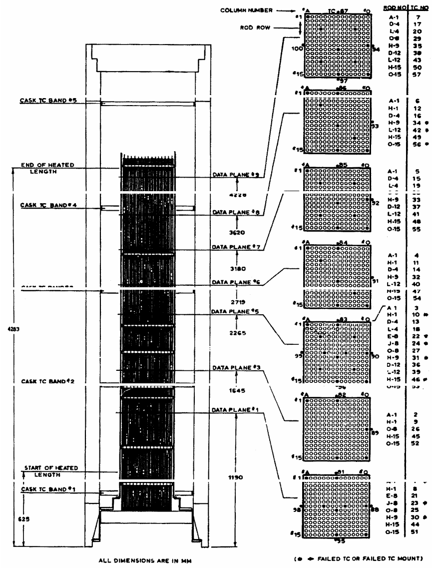

Figure 1. Schematic of the test section

3. Simulation of single assembly heat transfer test

2.1 Heat generation in fuel rods

The 15x15 fuel array consists of 225 fuel rods and 9 dummy rods which do not release any decay heat. The fuel rods are assumed as a solid structure generating heat source of 13590 W/m3. The diameter and active length of fuel rods are 10.67mm and 3.81m, respectively. So each fuel rod produces 4.63 W and the total decay heat is maintained as 1kW. Figure 2 shows the mesh formation which describes 15x15 subchannel constructed by Gambit 2 [2].

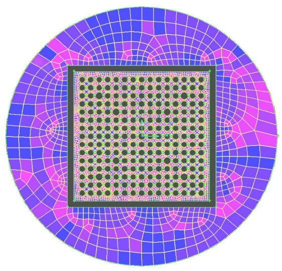

Figure 2. Mesh formation of 15x15 fuel assembly

2.2 Balance equations

It is assumed that the generated heat from nuclear pellet is conducted to cladding of fuel rods and then transferred to the environment by thermal convection and radiation on the solid surface (Eq. 1).

$$-kA\left.\frac{dT}{dr}\right|_{wall} = hA(T_{wall} - T_{\infty}) + F_G \varepsilon \sigma A (T_{wall}^4 - T_{surr}^4) \quad (1)$$

Thermal convection is produced by local differences between gravitational force and local fluid density variation. For incompressible fluid flow, Bossinesq approximation is available in steady state solution (Eq. 2) [2]. So it treats the density as a constant value in all solved equations except for the buoyancy in the momentum equations.

$$\beta = \frac{1}{v}\frac{\partial v}{\partial T} \quad (2)$$

$$Ra = Pr\, Gr \quad (3)$$

In the natural circulation, the heat transfer coefficient is a function of the Nusselt number which is a function of the Rayleigh number (Eq. 3) [3]. It is not easy for complex geometry to derive governing relationship expressed by the Rayleigh number.

The energy flux leaving a surface is composed of direct emitted and reflected energy when there is no scattering and no absorption in the medium (Eq. 4).

$$\varepsilon\sigma T_k^4 + \rho_k \dot{q}_{in,k} = \dot{q}_{out,k} \quad (4)$$

2.3 Simulation methodology

All fuel rods are treated as solid heat sources which generate the same quantity of experimental cases. To take into account thermal convection, gravity effect and laminar flow model are adopted. The density variation due to temperature change is neglected by Bossinesq approximation and the fluid density is assumed as a function of temperature. For radiation heat transfer model, the surface-to-surface model is selected in order to account for radiation exchange in an enclosure of gray-diffuse surface without optical thickness. The emissivity of fuel rods and other surfaces are considered as 0.8 and 0.3, respectively.

4. Results

The total power generated in fuel rods is transferred outside. The portion of the thermal radiation is 37% of the total power. Heat transfer through an inner plate in the canister is about 83% of the total power. So the inner wall of the inner plate receives less thermal radiative heat than the outer wall does.

The temperature distributions, which compared with experimental data, are shown in Figs. 3 and 4. The deviation between the maximum temperature in the CFD calculation and the data is around 5 °C.

There are some discrepancies in predicting the temperature distributions along the axial direction. But the axial temperature gradient is maintained.

Table 1. Heat balance sheet

Total power generated (total radiative power)	85.5 W (32 W)	100 % (37 %)
Total cooling (total radiative cooling)	85.3 W (42 W)	99.9 % (49 %)
Power through inner plate (radiative power on inner wall) (radiative power on outer wall)	71 W (23 W) (35 W)	83 % (32 %) (49 %)

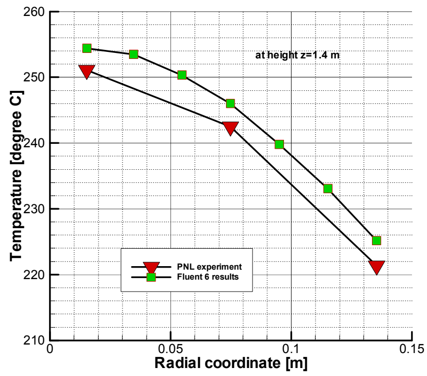

Figure 3. Radial temperature distribution

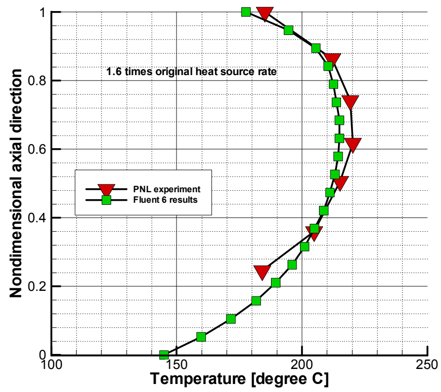

Figure 4. Axial temperature distribution

5. Conclusion

The temperature distributions and balance of heat transfer are obtained considering conduction, natural convection and thermal radiation. A methodology of heat transfer analysis in a spent fuel assembly is proposed by FLUENT 6. The deviation between the maximum temperature in the CFD calculation and the data is around 5°C.

REFERENCES

[1] COBRA-SFS, PNL Reports, Pacific Northwest National Laboratory, Richland, Washington, U.S.A. (1989)
[2] J.P. Holman, "Heat Transfer", 9th edition, Chapter 8, MacGraw-Hill (2002)
[3] Adrian Bejan, "Convection heat transfer", 3rd edition, Willey. (2004)