Transactions of the Korean Nuclear Society Autumn Meeting
Pyeongchang, Korea, October 30-31, 2014

Evaluation of thermal margin for the nuclear fuel bundle in the MAST assembly with Multi-scale analysis

Byong-Jo Yun [a*], Young Soo Kim [a], Hui-Yung Kim [a], Jun-Yean Lee [a], Jae-Yeong Jeon [b]
[a]*Mechanical Engineering Department, Pusan national Univ., Jangjeon-dong, Guemjeong-gu, Busan, 609-390, Korea*
[b]*POWER MnC Co., Ltd. Baengnok-ri, Habuk-myeon, Yangsan-si, Gyeongsangnam-do, 626-862, Korea*
[*]*Corresponding author: bjyun@pusan.ac.kr*

1. Introduction

MAST assembly is the key equipment of FTS (Fuel Transfer System) to keep and transfer one of the fuel bundle extracted at the nuclear reactor in the refueling process. MAST assembly which includes the spent fuel is moved in a pool filled with a coolant. At this moment, decay heat generated in the fuel is removed through the coolant of the pool. If the residual heat is not cooled effectively, a damage of the fuel is caused. Therefore, decay heat should be continuously transferred from fuel to coolant in the pool to ensure the integrity of the spent fuel during the refueling process. Also, Korea Institute of Nuclear Safety (KINS), domestic licensing organization, requires followings to prevent damage of nuclear fuel rod in the spent fuel-pool. In the accident condition spent fuel storage building 1) the fuel should be submerged under 3 m from water surface at least, and 2) coolant temperature in the pool should be maintained below 60℃ for safety of spent fuel [1].

In this study, we evaluated the integrity of the spent fuel under the above requirement in postulated accident condition. Thermal margin of the spent fuel bundle can be quantified by M-DNBR. M-DNBR is defined as the ratio of released heat flux at heated surface and CHF is expressed as follows,

$$MDNBR = \frac{q''_{CHF}}{q''_{w}} \quad (1)$$

To obtain the M-DNBR, we apply multi-scale analysis which consists of CFD analysis by STAR-CCM+ for the investigation of 3-dimensional flow behavior and 1-dimensional safety analysis code by MARS for the calculation of CHF.

2. Natural convection analysis with CFD code

2.1 Preparation of calculation mesh

MAST assembly shown in Fig. 1 is composed of MAST, Hoist box and fuel bundle. Fuel bundle is inserted in Hoist box and these are installed to MAST for the transportation. In the present study, 16 X 16 fuel bundle for the Advanced Power Reactor 1400 (APR1400) is chosen and simplified for the investigation.

Configuration of fuel bundle and its dimensions are presented in Fig. 2.

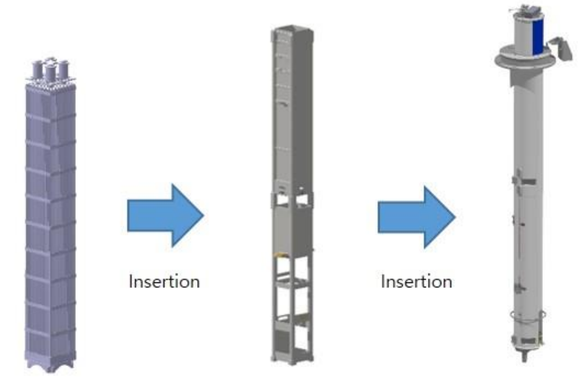

Fig. 1. Composition of MAST assembly

As shown in the figure, diameter of each fuel rod is 9.5 mm and pitch of the fuel assembly is 12.85 mm. Heated length of the fuel rods is 3810 mm. All of the fuel assemblies have many component for fixing the fuel rods and mixing coolant. However, most of this components act as flow resistance in the natural convection. Due to this factor, inlet velocity will be decreased. In the modeling of this element, a great number of meshes and then excessive analysis times are required. Moreover, precise meshes for the spacer grid affect significantly stability of numerical calculation. Therefore, in the present study, we assumed that nuclear fuel bundle is only composed of fuel rods as shown in Fig. 2.

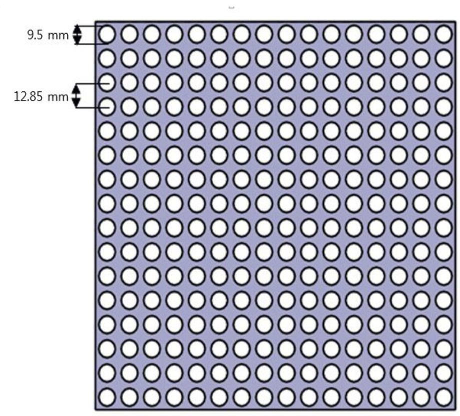

Fig. 2 Simplified geometry of Fuel bundle

Here, the reduction of the pressure loss that occurred with assumption about bundle geometry will cause an increment of inlet velocity. To compensate this effect, we tried to use the minimum velocity calculated from the CFD when we calculate CHF.

The MAST includes camera that indicates the work status to operator and the others. However, it is located at the top of MAST assembly, and thus it is also not modeled in the present work. And, the total axial length of the MAST assembly is about 10000 mm, however, lower of assembly of which length is 6810 mm is simulated because remaining part is located above the water level, as shown in Fig. 3.

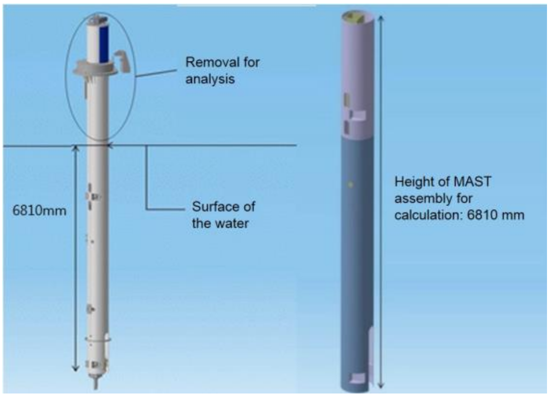

Fig. 3. Simplified geometry of MAST assembly

2.2 Numerical domain

In case of natural convection, we can qualitatively predict flow filed around MAST assembly as shown in the Fig. 4. Additionally, we should take account of external flow near the MAST assembly carefully because it affect the size of numerical domain for the pool.

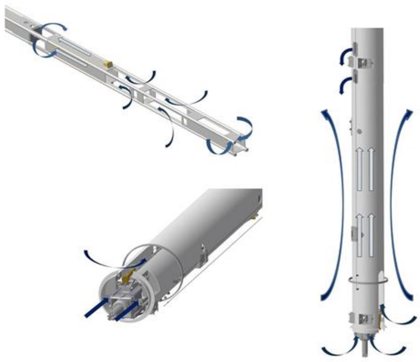
Fig. 4. Expected flow path around MAST assembly

In order to determine the calculation domain size, we conducted preliminary analysis with changing size of calculation domain. First, we added the fluid region of 400mm at the bottom to allow inflow of recirculation into the entrance of the MAST assembly. And then we repeated the flow analysis by changing pool. From this parametric study, it found that the optimum size to maintain the external flow is 1500 mm X 1500 mm. Fig. 5 shows flow field obtained from this calculation. Finally, the size of numerical domain was determined with 1500 mm X 1500 mm X 7210 mm

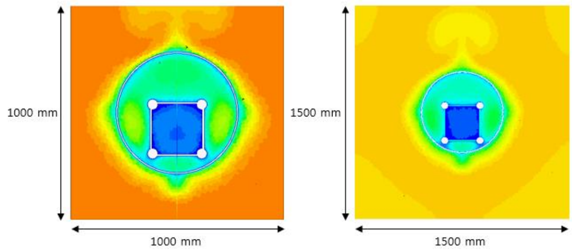

Fig. 5. Comparison of preliminary flow analysis 1000 X 1000(left), 1500 X 1500(right)

2.3 Mesh generation

We simplified geometry for the reduction of mesh number. However, due to the narrow spacing between the fuel rods, a number of meshes are still excessively generated. Because of this reason, we divided the numerical domain into two separated regions and assigned mesh conditions separately. One of the regions is fuel region which includes sub-channel of fuel bundle, and the other is fluid region which consist of sub-channel in the MAST assembly and pool. Finally, meshes of about 13 millions are generated.

2.3 Physics model and boundary condition.

Main physics models that are considered for the adiabatic flow analysis are listed in Table 1. For natural convection analysis, we additionally selected physical models to treat buoyancy effect caused by temperature variation. They are fluid temperature, gravity, and Boussinesq model.

Table 1 : Physics models

Type	Physics
Space	Three dimensional
Time	Steady-state
Material	Liquid
Flow	Segregated flow
Equation of state	Constant density
Viscous Regime	Turbulent
Turbulence	Standard K-Epsilon Low-Re
Wall Treatment	Low y+ wall treatment

Turbulence model is one of prime importance for the accurate prediction of natural convection flow. In this analysis, Standard K-epsilon Low-Re turbulence model which developed by Lien was adopted. In general, Standard K-epsilon turbulence model is limited to range of low Reynolds number, rotating flow, strong adverse pressure gradients & recirculation region and non-circular duct. However, this turbulence model used for this analysis is available for low Reynolds number through the damping function [2, 3]. Therefore we applied the turbulence model as in the present study.

The natural convection analysis needs heat flux on fuel rods. The heat flux was determined by following guidance of KINS which is Korean regulatory organization [4]. That is, the decay heat for the safety analysis is corresponding to that at 150 hours after reactor shutdown. In the present study, the total decay heat for the simulation is 52.3 kW for a fuel bundle and thus, uniform heat flux with 1796.65 W/m^2.is applied [5].

2.4 Results of adiabatic flow analysis

Adiabatic flow analysis was performed to confirm the soundness of mesh models between fluid region and fuel region before performing the natural convection analysis. Furthermore, to determine the meshing condition, we analyzed the mesh sensitivity in terms of numerical convergence. Fig. 8 represents the flow contour of X-Z plane. In this figure, the enlarged portion indicated that the flow along longitudinal direction is continuously calculated without any clogging. Radial flow is also the same.

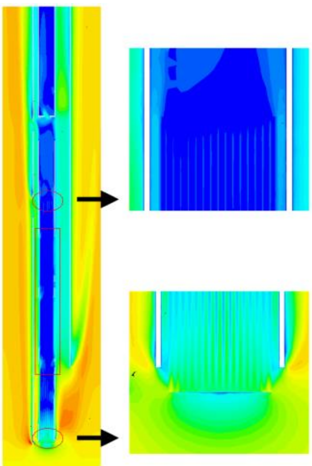

Fig. 6. Flow contour of adiabatic flow

2.5 Results of natural convection analysis

The analysis was performed by adding heat flux boundary conditions to the adiabatic one to simulate natural convection flow.

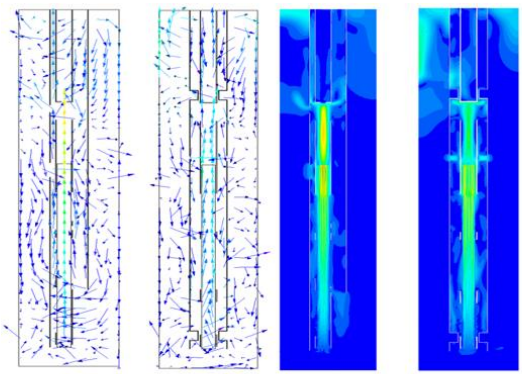

Fig. 7. Comparison of velocity field

Fig. 9 shows the calculated flow field and velocity vector of the each sectional planes. As shown in the figure, we were able to find out upward flow behaviors in the inside of the MAST assembly and downward flow at the pool. This was similar to the flow field that has been expected in Fig. 4.From this calculation, we need to obtain the inlet velocity at the bottom of MAST assembly for providing as a boundary condition of MARS. The entrance region of the MAST assembly for the determination of inlet velocity is appeared in the Fig. 10.

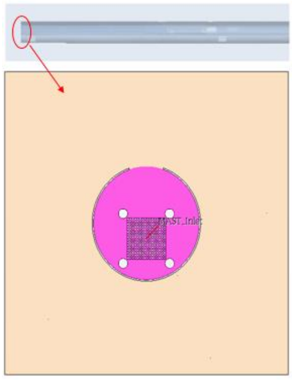

Fig. 8. Location of inlet and cross section

In the calculation, the minimum coolant velocity was found 2.78 mm/s at the inlet region. Additionally, Fig. 11 presents the temperature distribution along the fuel rods and pool center. It is also observed that the maximum fluid temperature of pool reached to 60.8℃. And we also calculated volume averaged temperature of the fluid in the all of the pool regions. Its value was 52.7 ℃ satisfying requirement by regulatory body.

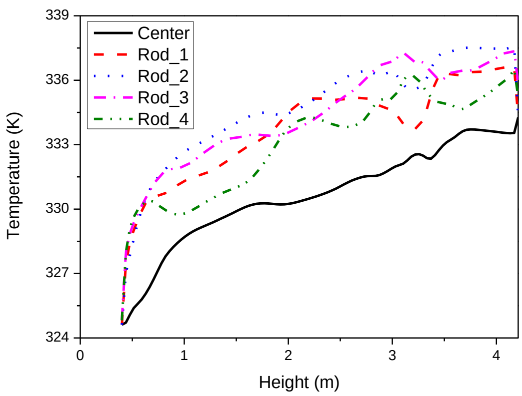

Fig. 9. Temperature distribution of the fuel rod and pool center

3. 1-D NPP safety analysis for the quantification of thermal margin

3.1 Input model for MARS code

We applied the minimum inlet velocity calculated by CFD code as a boundary condition for MARS code and then calculated CHF at a given flow condition. Input geometry for the MARS code is prepared by simplification of complicated geometry as in the CFD calculation. Nodalization for the MARS code is shown in Fig. 12. Flow path inside of MAST assembly was modeled by one dimensional pipe. To consider the pressure drop and flow characteristics at the sub-channel, we applied Rod bundle interphase friction model. And we modeled the fuel rods with a heat structure for heat transfer between fuel and fluid [6].

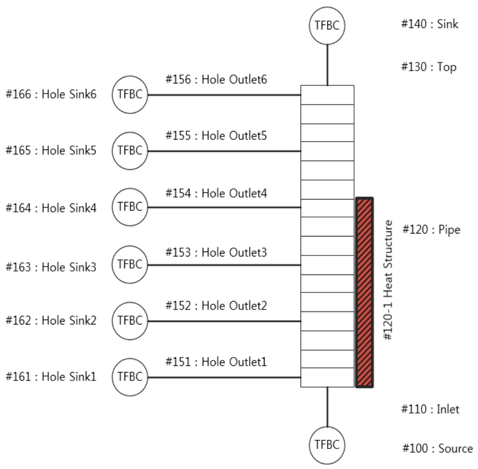

Fig. 12. Nodalization of forced convection input

3.2 Results of MARS analysis and quantification of thermal margin

We calculated the CHF by using the forced convection heat transfer analysis by the MARS code. From the calculation, CHF occurs when void fraction reaches to 0.75 as shown in Fig. 13. In the present flow condition, the CHF value was predicted with the heat flux of 489.08 kW/m2. Finally M-DNBR was found 272 which is a ratio of calculated CHF to applied heat flux. It is notable that the typical design limit of M-DNBR for the operating reactor is 1.3. It confirms that the spent fuel rods inside of the MAST assembly have a sufficient thermal margin in a postulated accident condition.

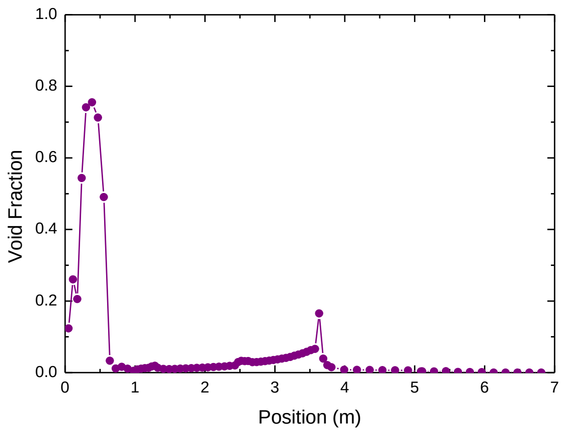

Fig. 13. Void fraction distribution along the fuel rod in the CHF condition

4. Conclusions

In the present study, we performed multi-scale safety analysis with combination of 3D CFD and 1D system codes in a postulated accident condition of MAST assembly. The CFD analysis was carried out for the confirmation whether stable convection occurs or not in the spent fuel assembly region. The 1D safety analysis with MARS code was also performed to obtain CHF and then quantification of M-DNBR. The analysis showed that the M-DNBR is 272 in an accident condition. It confirms that the spent fuel rods inside of the MAST assembly have a sufficient thermal margin in a postulated accident condition.

ACKNOWLEDGEMENTS

This work was supported by Nuclear Research & Development Program of the NRF (National Research Foundation of Korea) grant funded by the MSIP(Ministry of Science, ICT and Future Planning) and by the Nuclear Safety Research Center Program of the KORSAFe grant funded by Nuclear Safety and Security Commission (NSSC) of the Korean government (Grant code: NRF-2012M2A8A4055548, 1305011).

NOMENCLATURE

q’’	heat flux [W/m2]

Subscripts

CHF	critical heat flux
W	heated wall

REFERENCES

[1] S.H. Ryu, 2009, Cooling and purification system for spent fuel storage, Safety Review Guidelines for Light Water Reactors, Chap.9.1.3, pp.9~10.
[2] CD-adapco, 2014, Using K-Epsilon Turbulence, STAR-CCM+ User guide 9.02, pp 3392~.3396.
[3] Lien, F.S., Chen, W.L., and Leschziner, M. A., 1996, Low-Reynolds number eddy-viscosity modelling based on non-linear stress-strain/vorticity relations, Proc. 3rd Symp. On Engineering.
[4] Private communication with KINS.
[5] Neil E. Todreas and Mujid S. Kazimi, 1990, Nuclear Systems 1, Taylor & Francis, New York, pp. 66~67.
[6] Bayless, Paul D., 2011, RELAP5/MOD3. 2 Assessment Using CHF Data from the KS-1 and V-200 Experiment Facilities. INEEL/EXT-01-00782, Idaho National Engineering and Environmental Laboratory (July 2001).