CFD Simulation for Flow Behavior in Boiling Water Reactor Vessel and Upper Pool under Decommissioning Condition

Y. T. Ku, S. W. Chen, J. R. Wang, C. Shih, Y. F. Chang

***Abstract*—**In order to respond the policy decision of non-nuclear homes, Tai Power Company (TPC) will provide the decommissioning project of Kuosheng Nuclear power plant (KSNPP) to meet the regulatory requirement in near future. In this study, the computational fluid dynamics (CFD) methodology has been employed to develop a flow prediction model for boiling water reactor (BWR) with upper pool under decommissioning stage. The model can be utilized to investigate the flow behavior as the vessel combined with upper pool and continuity cooling system. At normal operating condition, different parameters are obtained for the full fluid area, including velocity, mass flow, and mixing phenomenon in the reactor pressure vessel (RPV) and upper pool. Through the efforts of the study, an integrated simulation model will be developed for flow field analysis of decommissioning KSNPP under normal operating condition. It can be expected that a basis result for future analysis application of TPC can be provide from this study.

***Keywords*—**CFD, BWR, decommissioning, upper pool.

I. Introduction

AFTER years of research and improvement, there is an apparent advancement in analysis skill of nuclear power plant (NPP). In the past, only a simply estimate of entire domain analysis could be calculated. With the development of CFD, more thermal-hydraulic characteristics and their effects on the system have been discovered and rapidly improved. The institution of national research, Institute of Nuclear Energy Research (INER), was utilized CFD calculation in system analysis of NPP. The cases of system analysis had been simulated by CFD including dry storage of spent fuel [1], alternative shutdown cooling behaviors for Chinshan NPP [2], and thermal-hydraulic analysis in vessel of BWR [3]-[5]. The researcher could provide better ways to improve the practical problems of cases could be provided by CFD results.

In Taiwan, according to the government policy, all NPPs were launched decommissioning plan in the next few years. However, the SFP of KSNPP was nearly full. Some operating fuel in reactor core could not be transferred to SFP when the decommissioning plan was started. There was insufficient capacity of SFP for all operating fuel. Therefore, INER conceived a modified project of upper pool. In this construct, three groups of rack were placed in upper pool, and a space could be provided for about 221 fuel assemblies. Considering that the upper pool was not designed for spent fuel storage, it was needed to estimate the feasibility of this project.

In this research, the purpose was focused on the analysis of flow behavior and mixing phenomenon in vessel and upper pool of KSNPP by using CFD – FLUENT [6]. The CFD model under normal storage situation was simulated and analyzed in this study.

Y. T. Ku, S. W. Chen are with the Department of Engineering and System Science, National Tsing-Hua University, R.O.C., Taiwan (e-mail: yutingku1102@gmail.com).

J. R. Wang, C. Shih, Y. F. Chang are with the Institute of Nuclear Engineering and Science, National Tsing-Hua University, and Nuclear and New Energy Education and Research Foundation, R.O.C., Taiwan.

II. CFD Modeling

Since FLUENT was utilized for past studies [1]-[5], sufficient experience of code application was assisted to construct the analysis methodology. FLUENT was chosen to develop the 3D model of KSNPP in this study and flow distributions in vessel and upper pool were obtained by simulated results. The modeling process, mesh constructed, and simulation conditions set up were described at following sections.

A. Geometry Model and Simplification

The CFD model was constructed based on engineering graphics of KSNPP. In this study, the upper plenum was empty due to steam generator, and dryer was removed, and the flow field between upper pool and vessel was connected. Some appropriate assumptions and simplifications were carried out for reduced calculation difficulty, and the presently adopted assumptions and simplifications for the KSNPP were as follows:

1. There were no solid structures in this model.
2. The wall in this CFD model was considered as adiabatic for prevent heat loss while we calculated the flow mixing.
3. Complex geometry of fuel assembly was simplified as simple rectangle, but flow resistance was considered by adding a porous zone in order to approach actual flow field and pressure distribution under operating conditions.
4. Small components, gap between structures and heat capacity of structure itself were ignored due to the little effect on entire flow field.
5. For increase model functionality, geometry model was divided into five parts and the parts were fitted together by interface boundary system. Through this system, it was easy for one part to change model geometry or replace grid distributed without affecting other parts.

Hence, the CFD model was divided into five parts including downcomer region, lower-plenum region, core region, upper

International Journal of Chemical, Materials and Biomolecular Sciences
ISSN: 2415-6620
Vol:12, No:3, 2018

plenum and cavity region, upper pool and gate region. Data transportation between different parts was utilized interface boundary system. During calculation, two layers were designated at two face of neighbor parts, and the layers were selected as a pair for data transport in interface boundary system. Each parts of CFD model and corresponding interface denominations were shown in Fig. 1.

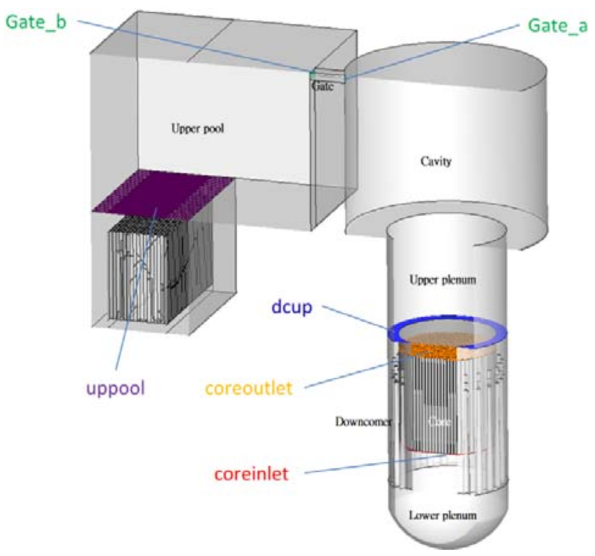

Fig. 1 Structure of each part and corresponding interface boundary layer location in CFD model

B. Mesh Generation and Sensitivity Study

In order to avoid large computational difference caused by incorrect mesh distribution, mesh sensitivity study was needed to maintain good mesh quality. Structured grid system was employed to generate majority meshes in this model except downcomer and lower plenum. Unstructured tetrahedron grid configuration was utilized to generate meshes in downcomer and lower plenum due to the complex geometry. Mesh skewness of entire model was controlled below 0.6, and maximum mesh skewness 0.8 was in a minority of meshes. The total grids were about 5.88 million, and the mesh configuration was shown in Fig. 2.

Structured grid system was employed to generate majority meshes in this model except downcomer and lower plenum region. Unstructured tetrahedron grid was utilized to generate meshes in downcomer and lower plenum region due to the complex geometry. Mesh skewness of entire model was controlled below 0.6, and maximum mesh skewness 0.8 was in a minority of meshes. The total grids were about 5.88 million and the mesh configuration was shown in Fig. 2. In order to avoid large computational difference caused by incorrect mesh distribution, mesh sensitivity study was needed to maintain good mesh quality. According to best practice guidelines (BPG) of CFD [7], three cases of different mesh number for steady state condition were performed in downcomer region. Pressure difference of downcomer region was selected for this analysis, and simulation result was shown in Table I. Mesh uncertainty could be quantified as 1.0078%. The result demonstrated that fine mesh had little discrepancies on result but spent more CPU time to obtain similar analysis. Considering the time costs and computer resource, the standard mesh was adopted for the following calculation.

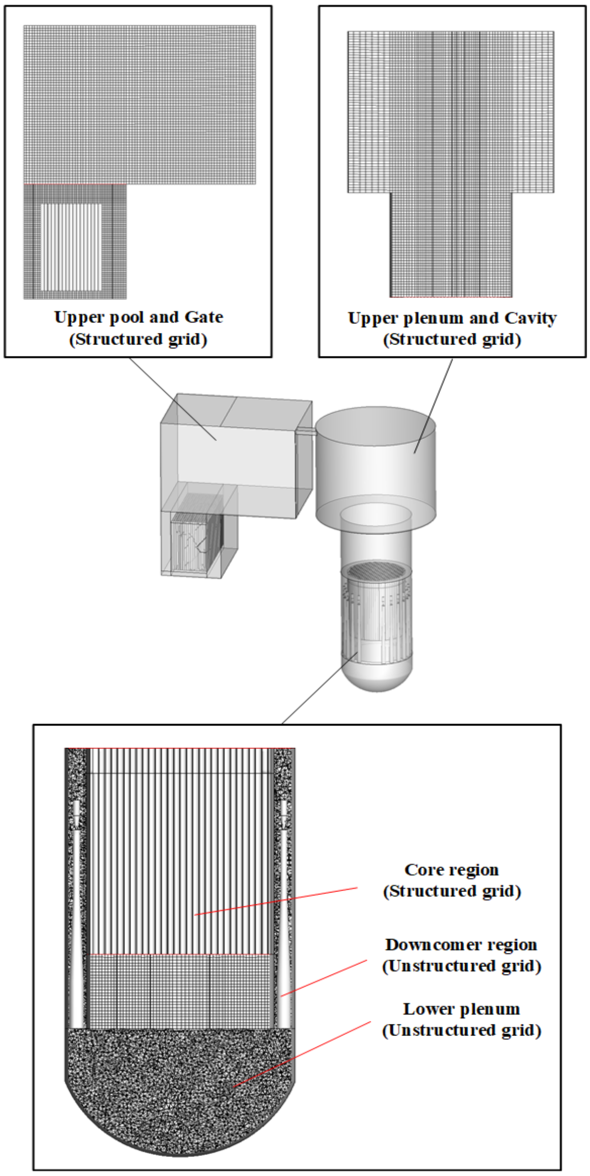

Fig. 2 Mesh distribution of different regions

TABLE I
SIMULATION RESULT OF DIFFERENT MESH NUMBER

Case	Mesh number	Pressure difference (Pascal)
Coarse mesh	532,486	5281.291
Standard mesh	1,865,648	5119.7549
Fine mesh	3,284,214	5104.5708

C. Simulation Method and Boundary Condition

Several methods were provided to solve the spatial discretization in FLUENT calculation including first-order upwind scheme, second-order upwind scheme, QUICK, etc. In

International Journal of Chemical, Materials and Biomolecular Sciences
ISSN: 2415-6620
Vol:12, No:3, 2018

this case, second-order upwind scheme was utilized to solve governing equations, and the convergent residues of energy, momentum and other equations were listed in Table II. In the solution of pressure-velocity coupling, semi implicit method for pressure linked equations consistent (SIMPLEC) [8] was selected.

TABLE II
SIMULATION METHOD SETTING

Pressure-Velocity Coupling	
Scheme	SIMPLEC
Spatial Discretization	
Gradient	Least Squares Cell Based
Pressure	Standard
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind
Convergent residues	Energy: 10^{-5} Others : 10^{-3}

Since the fuel assemblies were stayed in core for long-term cooling, this situation was similar to shutdown cooling mode of NPP outage. Therefore, the operating condition of shutdown cooling mode was used in previous study. When reactor was under shutdown cooling mode of NPP outage, cooling water circulation was carried out by RHR and FPCPS system. Table III showed the boundary conditions for input and output of RHR and FPCPS system under shutdown cooling mode.

TABLE III
INITIAL CONDITIONS OF SIMULATION

Initial condition	Mass flow rate (kg/s)
RHR system	
Jet-pump inlet	113.3579
RHR outlet	-113.3579
FPCPS system	
Diffusion tube inlet	75.57
FPCPS outlet	-75.57

III. RESULTS AND DISCUSSION

After model construction and analysis settings, obtaining and analyzing calculation results is the next step. Fig. 3 showed the velocity distribution of entire model. The overall velocity field in Fig. 3 was demonstrated that partition models were successfully connected by interface system and the trend of flow field was within expectations. In this case, cooling water was injected into jet-pump inlet by RHR system, and the fluid near inlet nozzle was sucked into jet-pump. The suction flow was mixed in jet-pump before infused into lower plenum. Fig. 4 presented more detail about suction effect and internal mixing phenomenon of jet-pump, and it could be seen that jet-pump function of this model was similar to reality situation. Flow direction was changed due to the bowl shape of lower plenum. An obvious vortex was found in Fig. 5 and caused flow speed slowdown to 0.08 m/s.

At core region, most fluid flowed passes through outside of fuel channel due to the flow resistance of fuel assembly. Part of fluid entered fuel channel and had higher velocity near core center. One-quarter view of the overall flow field in core region was shown in Fig. 6 and an obvious velocity gradient distribution was found at core bottom. Fig. 7 showed the working flow state in upper plenum. Flow speed drop and stagnation was occurred in this region since mixing space increased after removing steam separator and steam dryer. Before entering upper pool, working flow must pass through the gate connecting upper plenum and upper pool. However, the flow space had vehemently reduced from upper plenum to gate, and caused a rise in flow velocity. Finally, the working fluid was flowed in to upper pool, and entire flow field could be represented by Fig. 8. In the flow field of upper pool, cooling water was moved along wall and changed flow direction by model geometry. Flow speed drop and stagnation was also observed in this region. The maximum and minimum flow velocities of entire CFD model were listed at Table IV.

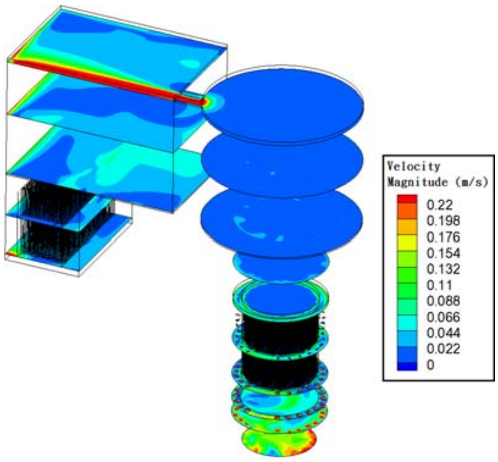

Fig. 3 Velocity distribution of entire model

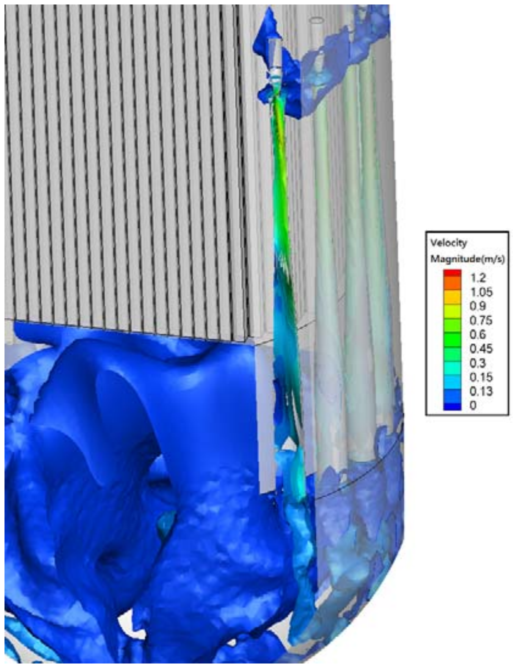

Fig. 4 Mixing phenomenon of jet-pump and lower plenum

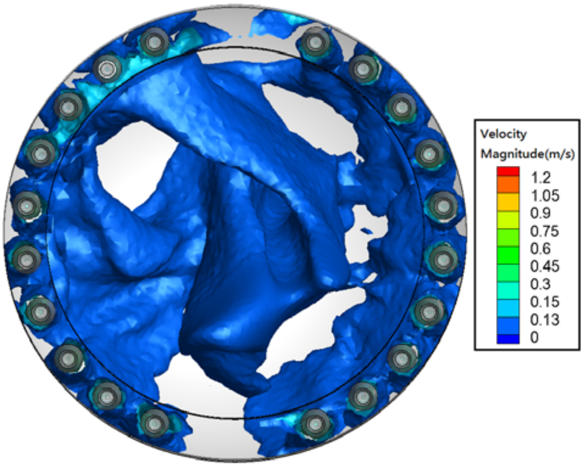

Fig. 5 Top view of flow field in lower plenum

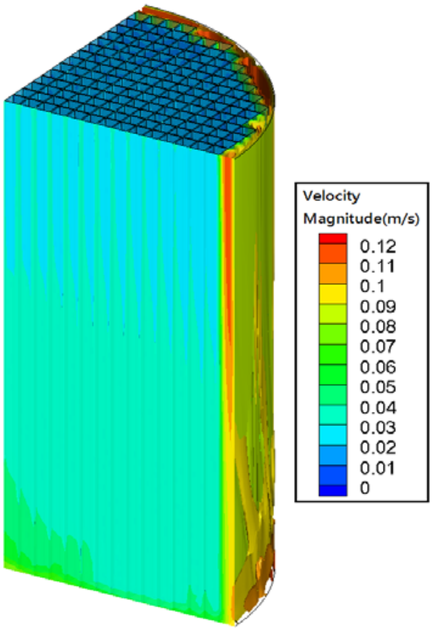

Fig. 6 Velocity distribution at core region

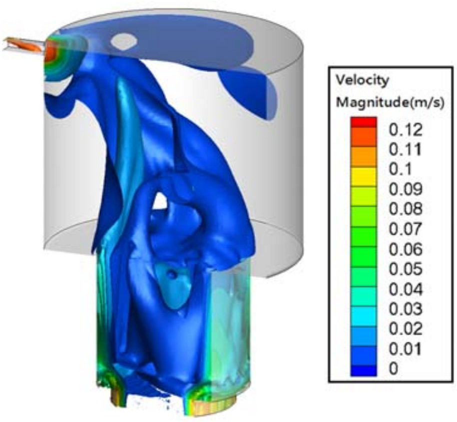

Fig. 7 Flow distribution in upper plenum and cavity

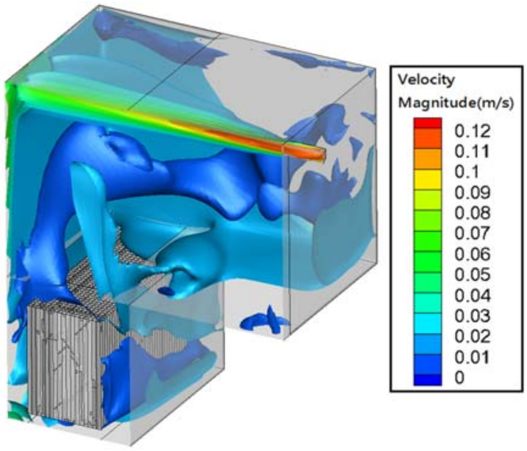

Fig. 8 Flow distribution in upper pool

TABLE IV
MAXIMUM AND MINIMUM VELOCITY IN DIFFERENT REGION

Region	Max. velocity(m/s)	Min. velocity(m/s)
Downcomer	0.73	2.56×10^{-4}
Jet-pump	3.97	2.55×10^{-3}
Lower plenum	0.74	1.071×10^{-3}
Core	0.13	8.00×10^{-5}
Upper plenum	0.13	2.00×10^{-4}
Cavity	0.33	1.41×10^{-4}
Gate	0.60	1.74×10^{-2}
Upper pool	0.57	2.09×10^{-6}

IV. CONCLUSIONS

In this study, a three-dimensional partial CFD model of Kuosheng pressure vessel and upper pool has been built to investigate mixing phenomenon, and the model was utilized to calculate the steady state condition under decommissioning situation. The mesh sensitivity study was completed for decided an appropriate mesh applying to the following simulation. Entire flow field under steady state was obtained, and several vortexes caused by complex geometry and flow direction changed were found by CFD results. The velocity differences at different region under decommissioning situation could be obtained. This study successfully demonstrates a CFD model for Kuosheng BWR pressure vessel and upper pool, and thermal analysis is the next objective. It can be expected that this model will be applied in nuclear reactor safety analysis.

REFERENCES

[1] Institute of Nuclear Energy Research, “Safety Evaluated Report: Thermal Analysis of Dry Storage and NPP Outage Units by CFD Methodology,” NRD-SER-99-06 (2011).

[2] Yung-Shin Tseng, Chih-Hung Lin, Yng-Ruey Yuann, Jong-Rong Wang and F. Peter Tsai, “Analyzing the Alternative Shutdown Cooling Behaviors for Chinshan Nuclear Power Plant Using CFD Simulation,” Annals of Nuclear Energy, Vol. 38 (11), pp.2557-2568 (2011).

[3] Yu-Ting Ku, Yung-Shin Tseng, Chih-Wei Su, Jong-Rong Wang, Chunkuan Shih, “The Thermal Hydraulic Behavior in Core Inlet Region of BWR with the Inadvertent Startup of HPCI,” NUTHOS-9, Kaohsiung, Taiwan (2012).

[4] Sahoshih Ma, Susen Hsu, Yung-Shin Tseng, “Evaluation of the Asymmetric Effects on Core Thermal Limits for the Inadvertent Cold Coolant Injection Transient in Chinshan BWR-4 Plant,” Annals of

Nuclear Energy, Volume 78, pp.15-25 (2015).
[5] Jan-Ru Tang, Chih-Chung Chung, Yung-Shin Tseng, "Analysis of the Inadvertent Start of HPCI Event for the Chinshan Nuclear Power Plant with CSAU Approach," EDSM2010-ICNMM2010, Montreal, Canada (2010).
[6] ANSYS, FLUENT V12 User's Guide, ANSYS Inc. (2009).
[7] J. Mahaffy, B. Chung, C. Song, et al., "Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications - Revision" Organization for Economic Co-Operation and Development, NEA/CSNI/R (2014)11.
[8] J. P. Van Doormaal, G. D. Raithby, "Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows," Numerical Heat Transfer, vol. 7, issue 2, pp.147-163 (1984).