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

A Preliminary Study of the Core Catcher System on Various Stud Shapes using FLUENT

Uiju Jeong, Gwang Hyeok Seo, Gyoodong Jeun, Sung Joong Kim*
Department of Nuclear Engineering, Hanyang University
222 Wangsimni-ro, Seongdong-gu, Seoul, 133-791, Republic of Korea
**Corresponding author: sungjkim@hanyang.ac.kr*

1. Introduction

In general, a nuclear reactor faces various potential thermal attacks from several heat sources including nuclear fission or decay reactions during the reactor operation. Under improper conditions, providing sufficient heat removal is considerably important to prevent undesirable events, such as design basis accidents (DBAs) or severe accidents [1]. Recently, a great interest in strategies for severe accident mitigation has been paid attention since the accident at the Fukushima Daiichi nuclear power plant occurred. For example, as a kind of in-vessel retention (IVR) strategies, reactor cavity flooding is used for Westinghouse's AP1000 and South Korea's OPR1000 [1,2]. Moreover, the European Pressurized Reactor (EPR) has adopted an ex-vessel core catcher strategy rather than the IVR strategy [3]. Although the mitigation strategies suggested are vigorously considered, there are still various issues due to its uncertainties and complex phenomena during severe accidents [4]. In this study, to assess the effect of studs installed on the core catcher body, a CFD analysis for coolant channels having rectangular or cylinder shaped studs is carried out.

2. Core Catcher System for Severe Accident Mitigation

The progression of severe accidents is classified as in-vessel and ex-vessel events generally in accordance with the structural integrity of a reactor vessel. To avoid its complex phenomena, which is hard to resolve, the IVR are considered with several strategies including the external reactor vessel cooling (ERVC) and installation of an in-vessel core catcher placed inside of the reactor lower head [5]. On the other hand, ex-vessel events could be expected if the vessel failure occurs [6]. As one of promising strategies, the ex-vessel core catcher systems are designed for retention and cooling of molten materials of nuclear fuels and structures, which is a significant heat source to attack and degrade the integrity of a reactor vessel. In the system, the molten materials spread on a large area, and experience more effective cooling by water flowing [3]. The ex-vessel core catcher system has been adopted in several nuclear reactors. For example, Tianwan plant, which is a Russian VVER-1000 type [7], adopted the system. Moreover, the ex-vessel core catcher system is also considered in a design stage for U.S. EPR [8] and APR1400 (South Korea) plants [9].

2.1 Description of Core Catcher System

Figure 1 shows an arrangement of the ex-vessel core catcher system of EU-ARP1400 with reactor vessel, cavity and IRWST [4,9]. The core catcher body is placed in the reactor cavity, and molten materials will be relocated to the body after the vessel failure.

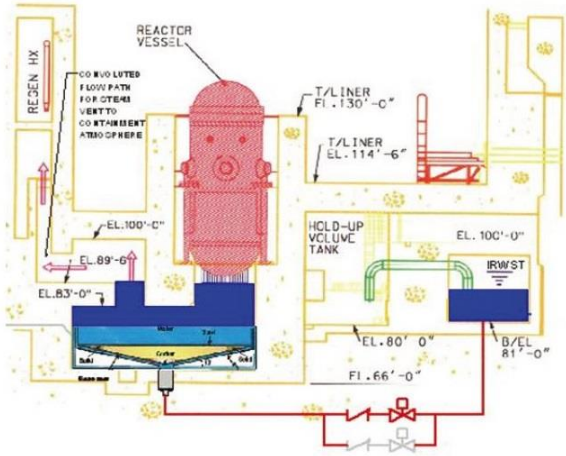


Fig. 1. Arrangement of RPV and ex-vessel core catcher system [4,9].

The core catcher system suggested for EU-APR1400 has a variety of features. Figure 2 shows a conceptual design of the core catcher system considered for EU-ARP1400 [4,9]. Once the molten materials move to the core catcher body, cooling water from the IRWST will be flooded to the downcomer by gravity. After that, carrying heat generated from the molten corium, the water flows from the inlet of the coolant channel to the outlet. The water flowing out of the outlet is eventually cooled down, then the cooling process is driven by the natural circulation, and maintained until a complete dissipation of the generated heat.

Moreover, to support the core catcher body, a lot of studs, which also provide the coolant path of the channel, are installed between the core catcher body and reactor cavity wall as shown Fig. 3 [4]. The cooling channel is inclined at an angle of about 10^o for steam venting. The shape of studs has a rectangular geometry. Design parameters including the stud shape are currently open for optimization.

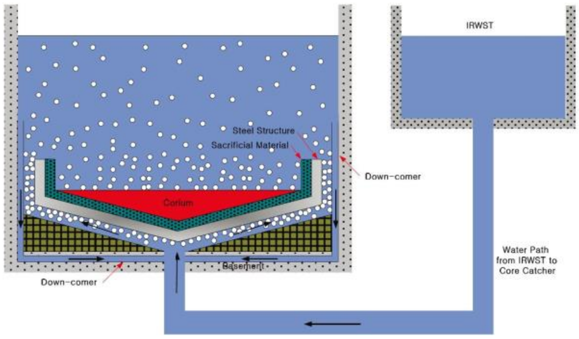


Fig. 2. Conceptual design of the core catcher system with natural circulation [9].

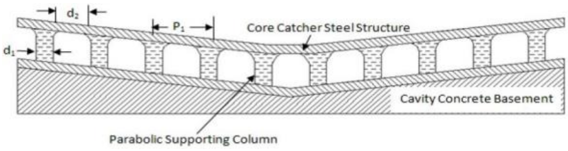


Fig. 3. Side view of the cooling channel [4].

2.2 The Effect of a Stud of the Core Catcher System

As described, the studs support the core catcher body and provide the coolant path. The coolant channel structurally maintained by the studs ensures continuous water flowing through the channel and heat transfer from the molten corium on the body surface.

However, an installation of studs could affect the flow of coolant carrying heat generated. The studs, in fact, are a kind of partial blockage against the flow. The streamline can be distorted by the blockages, and stagnation points may occur in the channel. Furthermore, locally degraded heat transfer around the points result in formation of vapor pocket. The vapor formed could also be developed as another obstacle for steam venting, which is one of current issues for the core catcher system [4]. Thus, it is important to find proper shapes of a stud, and design parameters are needed to be optimized.

In the current research, to assess the effect of a stud, several research stages are considered with numerical and experimental approaches. First, CFD simulation is carried out for a verification of streamlines and the amount of pressure drop with different stud shapes. Figure 4 shows suggested examples of the stud shape. Since rectangular stud, which is a basic type, is expected to distort the streamline greatly, circular or elliptic cylinder is also suggested. After reviewing results of CFD simulation, forced convective experiments for various coolant channels are conducted, especially to investigate local heat transfer. Moreover, MARS, the system safety analysis code, can also be utilized to predict void fraction at a certain heat flux applied to the channel. Finally, comprehensive analysis with the numerical and experimental results can help optimize design parameters on studs and the coolant channel. In this study, as a first step, a CFD analysis for coolant channels having rectangular or cylinder shaped studs is carried out.

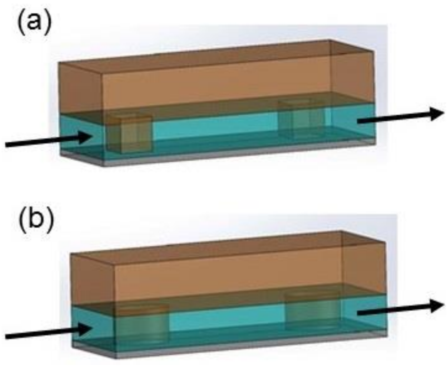


Fig. 4. Suggested examples of the stud shape with the coolant channel: (a) Rectangular stud; (b) Elliptic cylinder stud.

3. Numerical Modeling

In this study, a commercial CFD analysis tool, namely ANSYS FLUENT, was employed to simulate the hydraulic phenomenon in the core catcher system, especially the coolant channel containing short columnar structures called a stud. The effects of a stud shape on pressure drop and streamline in the coolant channel are major interests in this stage.

3.1 Modeling of the Coolant Channel

Figure 5 shows the geometry of the overall core catcher body [10]. Only a partial region of the coolant channel as indicated in Fig. 5 with dashed line has been numerically analyzed. Figure 6 shows the geometry of the modeled section. Eight studs are installed at the coolant channel to support the static and dynamic loading on the core catcher body. There are two types of stud: cylinder and rectangular shapes. Diameter of the cylinder shaped stud is 113 mm and width of the rectangular shaped stud is 100 mm.

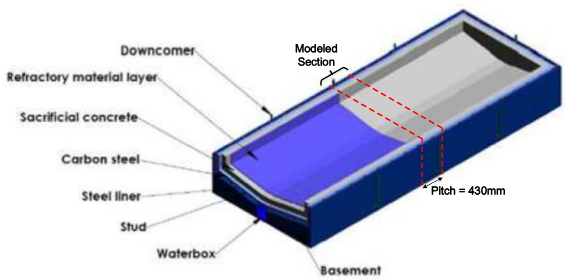


Fig. 5. Schematic diagram of EU-APR1400 Core Catcher [10].

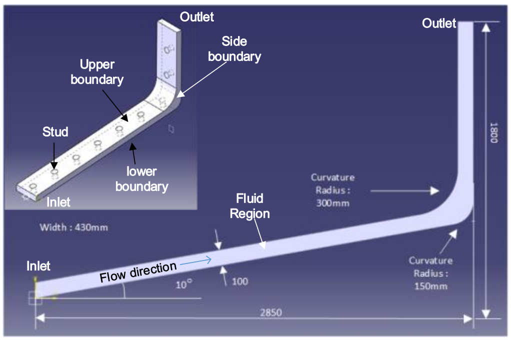


Fig. 6. Schematic diagram of the modeled section in the core catcher.

3.2 Grid Generation

The modeled section was meshed using ANSYS ICEM CFD program as shown in Fig. 7. To reduce the number of grids and increase the quality of the grids, only hexahedron type grids were utilized. The grid system generated contains about 9,000,000 nodes.

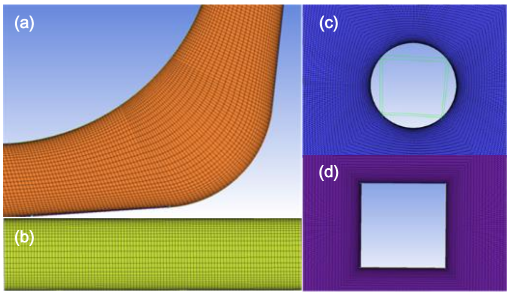


Fig. 7. Generated grids for (a) elbow section; (b) inlet; (c) surrounding of cylinder stud and (d) rectangular stud.

3.3 Solver Settings

After the meshing step, the mesh file generated was imported into FLUENT. Table I shows information about FLUENT solver settings used in the simulations. All the simulations were calculated under steady state and gravitational condition.

Table I. FLUENT Solver Setting

Inlet velocity [m/s] :		0.276
Turbulence model :		Realizable k-ε
Near-Wall Treatment :		Non-Equilibrium Wall function
Spatial Discretization	Gradient :	Green-Gauss Node based
	Pressure :	PRESTO!
	Momentum :	1st order upwind
	Turbulent kinetic energy :	1st order upwind
	Turbulent dissipation rate :	1st order upwind
Pressure-velocity coupling scheme :		SIMPLE
Operating temperature :		70 ^{0}C

Moreover, several initial and boundary conditions are considered. A constant velocity condition was imposed at the inlet. A no-slip condition was applied to the stud surfaces and the upper and lower boundaries. A symmetry condition was used at two side boundary surfaces.

4. Results and Discussion

ANSYS FLUENT enables us to analyze the motion of coolant in the coolant channel by solving numerically the governing equation. All the simulations continued until every residual falls below 1×10^{-4} and convergences of the residuals were shown up.

4.1 Flow Distribution

Figure 8 and 9 show the obtained velocity distribution in the middle plain containing coolant and the studs and streamlines. As shown, there is a tendency that low velocity region between studs develops as fluid goes downstream. This can be explained considering difference between the region near the studs and the side region where there is no stud. The difference leads more flow occurs around the region where lower flow resistance is expected. We could confirm that this phenomenon appears severely in the coolant channel containing rectangular shaped studs comparing Figs. 8 and 9.

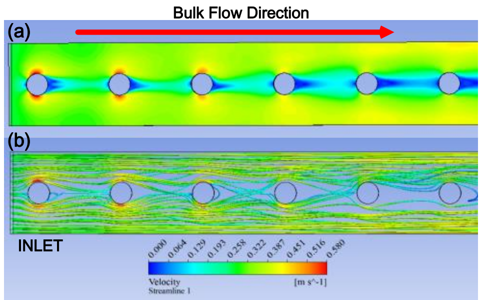


Fig. 8. Calculated flow distribution in the coolant channel containing cylinder shaped studs: (a) contour of velocity magnitude; (b) streamlines

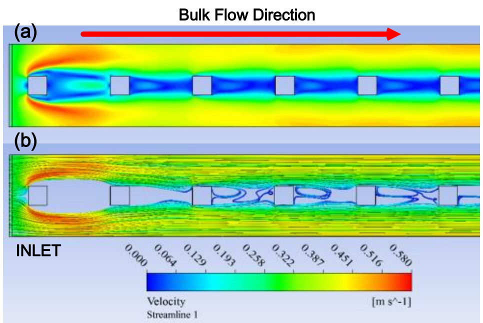
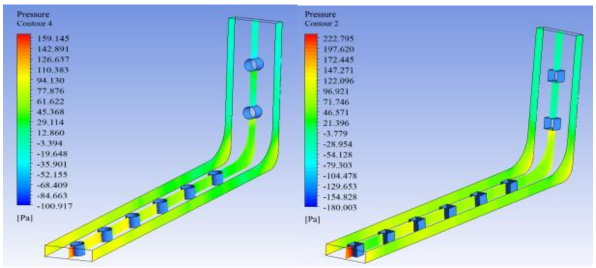


Fig. 9. Calculated flow distribution in the coolant channel containing rectangular shaped studs: (a) contour of velocity magnitude; (b) streamlines

4.2 Pressure Distribution

In the 4.1 chapter, we could confirm that there is a large velocity gradient near the studs. This means that the installed studs increases the pressure drop in the coolant channel significantly. And we could expect that the pressure drop in the channel containing rectangular shaped studs larger than that of cylinder shaped studs considering the extent to velocity gradients near the studs.

Figure 10 shows the calculated pressure contours in the channels. As expected previously, the calculated pressure drop in the channel containing rectangular shaped studs larger than that of cylinder shaped studs. The amount of pressure drop is 145 Pa for the rectangular shaped studs, and 100 Pa for the cylinder shaped studs.



(left) (right)

Fig. 10. Pressure contours in the coolant channel containing (Left) cylinder shaped studs; (right) rectangular shaped studs

5. Concluding Remarks

In this study, numerical simulations for the different stud shapes of the core catcher system were carried out using ANSYS FLUENT. For a comparison work, the rectangular and cylinder shaped stud were modeled with the same initial and boundary conditions. The major findings observed from this study can be summarized as follows.

- The simulation results showed the 31% reduced amount of pressure drop for the case of the cylinder shaped studs as compared with the reference case, which is for the rectangular studs.
- The tendency of reduced pressure drop is well in accord with the flow distribution. The fluid velocities around the studs were greatly distorted for the rectangular studs than those around the cylinder studs.
- The distorted stream of fluid could affect heat transfer from core catcher body, and result in locally additional damages. This result may suggest the necessity of finding an optimized stud shape.

For more improved comparison work, an additional simulation is planned including different stud shapes. Furthermore, for the next step, convective experiments and MARS code analysis are intended for a comprehensive analysis on finding the optimum shape of a stud to improve heat transfer capability of the core catcher system.

Acknowledgements

This work was supported by the National Research Foundation of Korea (NRF) grant funded by Ministry of Science, ICT and Future Planning (MSIP) with grant number, 2014M2B2A9032081.

REFERENCES

[1] S. Lee, K. S. Ha, H.-Y. Kim, and S. J. Kim, Validation of RCS depressurization strategy and core coolability map for independent scenarios of SBLOCA, SBO, and TLOFW, Journal of Nuclear Science and Technology, Vol.51, p.181, 2014.

[2] J. Buongiorno, L. W. Hu, G. Apostolakis, R. Hannink, T. Lucas, A. Chupin, A feasibility assessment of the use of nanofluids to enhance the in-vessel retention capability in light-water reactors, Nuclear Engineering and Design, Vol.239, p.941, 2009.

[3] M. Fischer, The severe accident mitigation concept and the design measures for core melt retention of the European Pressurized Reactor (EPR), Nuclear Engineering and Design, Vol.230, p.169, 2004.

[4] F.-B. Cheung, “Development of an Ex-Vessel Core Catcher for Eu-APR1400”, Korea Atomic Energy Research Institute, Daejeon, Korea 2010.

[5] J. H. Park, Y. H. Jeong, W.-P. Baek, S. H. Chang, An assessment methodology for in-vessel corium retention by external reactor vessel cooling during severe accidents in PWRs, Annals of Nuclear Energy, Vol.28, p.1237, 2001.

[6] H. Esmaili, and M. Khatib-Rahbar, “Analysis of In-Vessel Retention and Ex-Vessel Fuel Coolant Interaction for AP1000”, NUREG/CR-6849, ERI/NRC-04-201, U.S. Nuclear Regulatory Commission, Washington, DC 2004.

[7] L. V. Kukhtevich, V. V. Bezlepkin, V. F. Strizhov, V. B. Khabensky, V. B. Proklov, and Y. G. Leontieve, Severe Accident Management Measures for Tianwan NPP with WWER-1000, OECD Workshop on the Implementation of Severe Accident Management Measurements, Sept. 10-13, 2001, Paul Scherrer Institut, Villigen-PSI, Switzerland.

[8] AREVA NP Inc., “U.S. EPR Severe Accident Evaluation Topical Report”, ANP-10268NP, Lynchburg, VA, 2006.

[9] J. H. Song, H. Y. Kim, K. S. Ha, R. J. Park, J. T. Kim, and F.-B. Cheung, A Core Catcher Design for Advanced Light Water Reactors, Proceedings of ICAPP 2011, May 2-5, 2011, Nice, France.

[10] J. Suh, H. G. Kim, Development of Corium-Sacrificial Material Interaction Model for the EU-APR1400 Core Catcher, Proceedings of the KSME autumn meeting, Changwon, Korea, 2012.