Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 7-8, 2015

Experimental Study for Effects of the Stud shape of the Core Catcher System

Kyusang Song, Hong Hyun Son, Uiju Jeong, Gwang Hyeok Seo, Doyoung Shin, 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

As the number of nuclear power plants increases and their siting is in the vicinity of the highly populated area, the public are more concerned about nuclear safety. While the installation of the nuclear power plants renders enormous economic benefit, increasing of radiological risks to the public due to the hypothesized severe accident cannot be ruled out. In preparation of potential severe accidents, a nuclear power plant is equipped with diverse systems of engineering safety features or mitigation system dedicated to the severe accidents conditions. As a common strategy, a number of nuclear power plants adopt the in-vessel retention (IVR) and/or external reactor vessel cooling (ERVC) strategies. With the ERVC strategy, an additional system (core catcher system) to catch molten core penetrating the reactor pressure vessel (RPV) was proposed for advanced light water reactor [1].

The core catcher system is for Ex-vessel in the European Advanced Power Reactor 1400 (EU-APR1400) to acquire a European license certificate [2, 3]. It is to confine molten materials in the reactor cavity while keeping coolable geometry in case that the RPV failure occurs. The system consists of a carbon steel body, sacrificial material, protection material and engineered cooling channel. As shown in Fig 1, the engineered cooling channel of the ex-vessel core catcher was adopted to remove sensible heat and decay heat of the molten corium using cooling water flooded from the In-Containment Refueling Water Storage Tank (IRWST) by gravity [1]. A large number of studs are placed in the cooling channel to support the core catcher body [4]. While installation of the studs is unavoidable, the studs tend to interfere in the smooth streamline of the core catcher channel. The distorted streamline could affect the temperature distribution and overall coolability of the system. Thus, it is of importance to investigate the effects of studs on the coolability of the core catcher system.

In the current research, to evaluate the effect of a stud on the streamline and natural convective boiling performance, numerical and experimental approaches were taken. As a part of numerical approach, CFD simulation using ANSYS/FLUENT was carried out. The objective was to predict disturbance of the streamline and temperature distribution due to the interference of the studs.

Through the CFD analysis, it was found that installation of studs affects the streamline by partially blocking the channel. In addition, it was observed that stagnant flow exists at the back of the studs. Such disturbances were expected to differ with the shape of the studs. Thus various stud shapes of rectangular, cylinder, and ellipse were investigated. With this preliminary CFD study, flow boiling experiments were designed and conducted to investigate the effects of the different stud shape on the critical heat flux (CHF). The occurrence of the CHF is anticipated at the back side of the stud due to the possible existence of the hot spot driven by the accumulation of the vapors. Final objective is, therefore, to confirm the effect of the core catcher stud on the performance of boiling heat transfer and to assure that the designed core catcher system may work as intended.

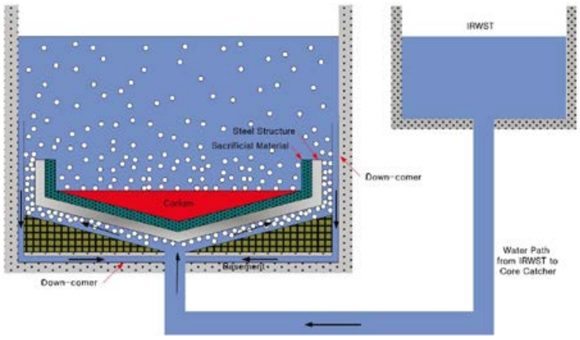

Fig 1. Cooling the conceptual design of the core catcher system [1].

2. Numerical method

To design the test section, ANSYS FLUENT, the commercial CFD analysis software, was employed to examine the possible hydraulic phenomenon in the core catcher system. The CFD analysis provides detailed predictions of streamline and temperature distribution in the coolant channel. In this section, detailed CFD methodology is addressed.

2.1 Modeling of the Coolant Channel

The core catcher is in a rectangular shape with a dimension of 6m × 16m and is geometrically symmetric along a longer axis as shown in Fig. 2 [5]. A dashed line indicates a single channel of the total coolant channel. Figure 3 shows the geometry of the single channel [6]. The test section is modeled to depict a partial region of the original coolant channel as indicated in Fig. 4, and scaled down by a factor of 1/10. The dimension of the flow channel including three studs is 129×43×10 mm^3 in length, width, and height respectively. The cooling channel also is inclined at an angle of 10° to facilitate steam venting during the cooling process [7]. There are three types of the stud shape: rectangular, cylinder and elliptic. Based on the blockage ratio, each diameter was determined as 10 mm since the reference square stud is 10 mm in each side.

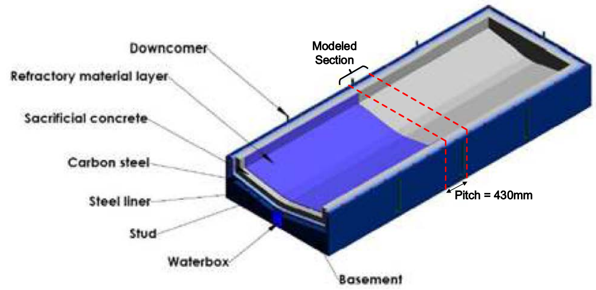

Fig. 2. Schematic diagram of EU-APR1400 Core Catcher [5].

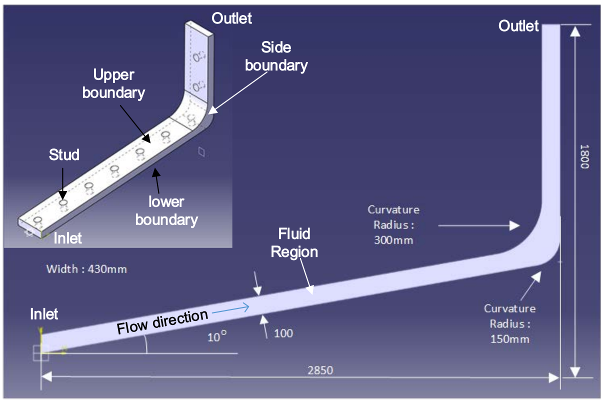

Fig. 3. Schematic diagram of the single coolant channel [6].

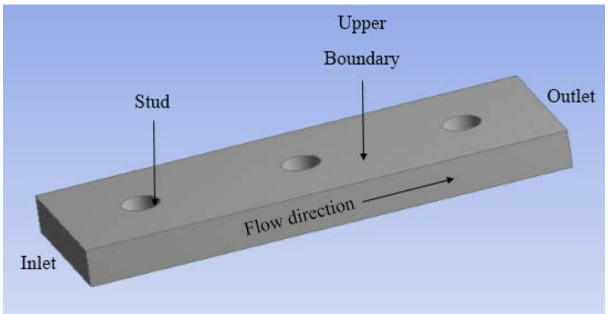

Fig 4. Schematic diagram of the test section in the core catcher.

2.2 Mesh Generation and Solver Settings

The test section was meshed using ANSYS workbench program as shown in Fig. 5. The mesh method adopts tetrahedron type of patch to confirm suitable generation of the grids while considering boundary conditions. The grid was generated with approximately 1,000,000 elements on each test sections.

After meshing, the generated mesh file was imported into FLUENT. All the simulations were calculated under steady state. The detailed information about FLUENT solver settings was presented in Table I. At the inlet, a constant velocity condition was applied. No-slip condition was applied to the stud surface, the upper and lower boundaries and two sides of boundary surfaces.

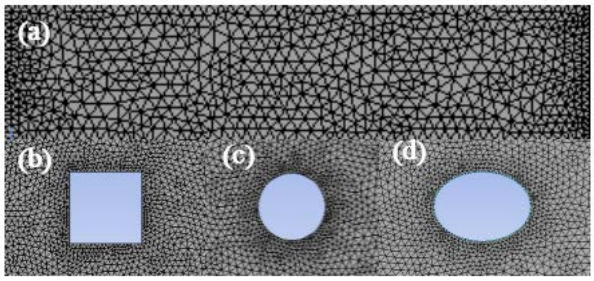

Fig. 5. Generated meshes for (a) inlet; (b) surrounding of rectangular stud (c) cylinder stud and (d) elliptic stud.

Table I. Solver Setting

	Inlet velocity [m/s] :	0.3
	Turbulence model :	Realizable k-ε
	Near-Wall Treatment :	Enhancement Wall Treatment
Spatial Discretization	Gradient :	Green-Gauss Node based
	Pressure :	Body Force Weighed
	Momentum :	QUICK
	Turbulent kinetic energy :	1st order upwind
	Turbulent dissipation rate :	1st order upwind
	Pressure-velocity coupling scheme :	SIMPLE

2.3 Calculation Results

ANSYS FLUENT helps to interpret the flow distribution in the coolant channel by solving the governing equations. In order to estimate the convergences, sufficient number of iterations was performed until each residual falls below 1×10^{-5}. The contours of velocity magnitude and streamline in the middle plain on the stud shapes; rectangular, cylinder and elliptic are shown in Fig. 6. As shown in Fig. 6, each modeled sections show a tendency that low velocity region between studs exists when fluid moves along the cooling channel. It is expected that the studs are likely to hinder the smooth flow of the core catcher channel.

Through the CFD analysis, it could be confirmed that different flow patterns are generated with different stud shapes. Especially, flow disturbance in the low velocity region at the back of the rectangular shaped stud was greater than in the other stud shapes.

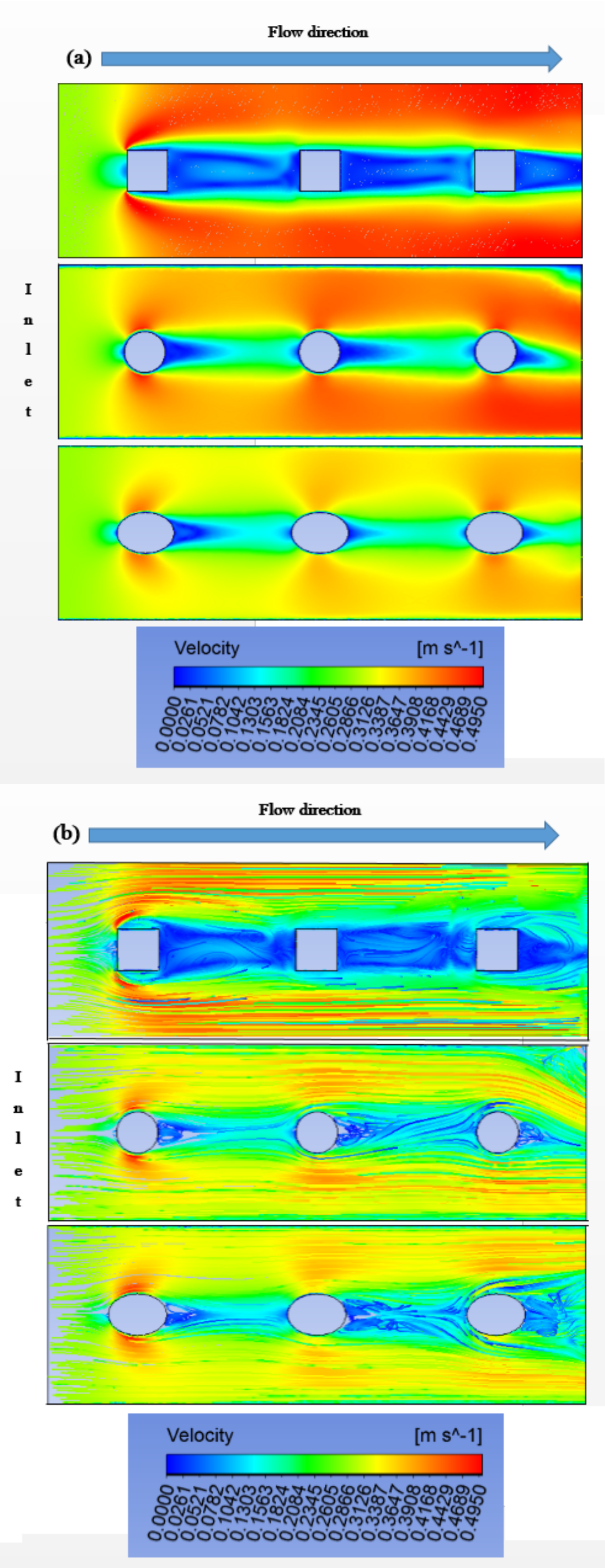

Fig. 6. Calculated flow distribution in the coolant channel containing rectangular, cylinder and elliptic shaped studs: (a) contour of velocity magnitude (b) streamlines.

3. Experimental Preparation

3.1 Experimental apparatus

The flow boiling heat transfer experiment was conducted with subcooled deionized (DI) water in the atmospheric pressure to investigate the effects of the different stud shapes on the boiling heat transfer performance. Figure 7 represents a schematic diagram of flow boiling heat transfer apparatus, which consists of a heat exchanger, a buffer tank, a centrifugal pump, a pre-heater, a flow meter, a DC power supply with allowable maximum capacity of 30V-2500A, a high speed camera, and data acquisition system. The schematic diagram of the test section is shown in Fig. 8. Heating surface made of stainless steel grade 304 (SS304), was installed in insulation material of 10-mm thick teflon block. Each end side of the heater is connected with copper electrodes and directly heated using the DC power supply. For the purpose of boiling visualization, flow window made of acryl was built in aluminum housing.

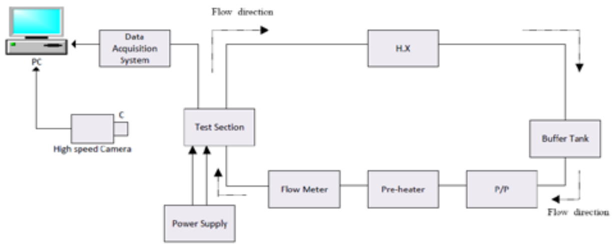

Fig. 7. Schematic diagram of flow boiling heat transfer apparatus

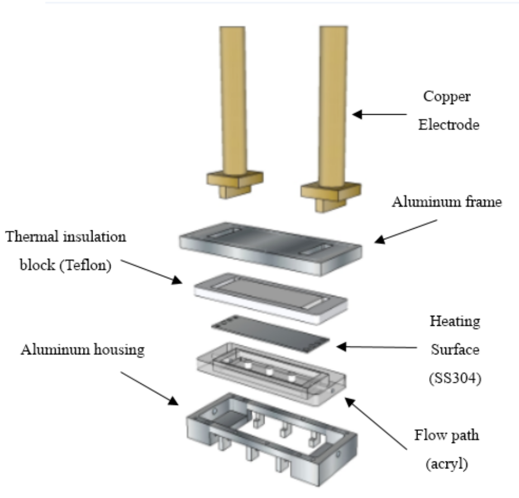

Fig. 8. Schematic diagram of flow boiling test section

Experimental procedure follows two major steps. First, inlet temperature of 97°C at atmospheric pressure was maintained using the preheater equipped in the apparatus because the IRWST is initially filled with water at a temperature of 97ºC [8]. This condition was continued until the end of the experiment. Second, power input was increased in step of 40 kW/m^2 until the heat flux reached 70% of the predicted CHF. Closed to the CHF, smaller heat flux of 20 kW/m^2 was applied to prevent sudden vapor film formation

3.2 The planning of the Experiment

On the basis of the CFD analysis, flow experiments are conducted to examine the hydraulic phenomenon on each stud shapes in Fig. 9. It could be confirm that different flow patterns are generated with different stud shapes. Since the hydraulic test, flow boiling experiments will be conducted to investigate the effects of the different stud shape on the CHF. The occurrence of the CHF is anticipated at the back side of the stud. K-type thermocouples are installed to measure the temperature at the back of the stud.

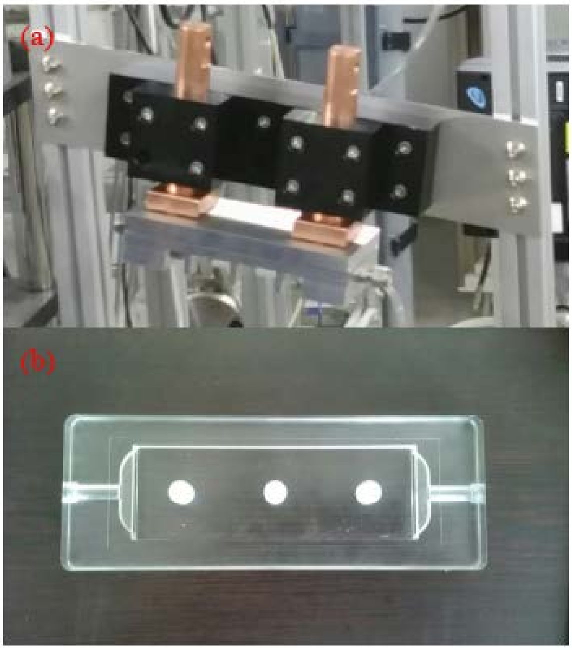

Fig. 9. Photo of (a) test section (b) flow path

4. Summary and Concluding Remarks

In this study, CFD simulations for the different stud shapes of the core catcher system were carried out using ANSYS FLUENNT. With this preliminary CFD study, flow boiling experiments were designed. The major findings observed from this study and future works can be summarized as follows.

- The CFD simulation results showed flow disturbance in the low velocity region at the back of the rectangular shaped stud was greater than in the other stud shapes.
- With the CFD analysis, the hydraulic test are conducted to confirm the different flow patterns with the shape of the studs.
- Since the hydraulic test, flow boiling experiments will be conducted to investigate the effects of the different stud shape on the CHF.
- Final objective is to confirm the effect of the core catcher stud on the performance of boiling heat transfer and to assure that the designed core catcher system may work.

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] 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.

[2] EUR (European Utility Requirements) for LWR Nuclear Power Plants, Revision C, April 2001.

[3] Y. S. Kim and J. H. Lee, "European Nuclear Design Requirements and EU-APR1400 Design Characteristics, "Proceedings of the KSME 2010 Fall Annual Meeting, Jeju, Korea, 2-5 Nov.2010

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

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

[6] U. J. Jeong, G. H. Seo, G. D. Jeun, S. J. Kim, A Preliminary Study of the Core Catcher System on Various Stud Shapes using FLUENT, Korea Nuclear Society Autumn Meeting, Pyeongchang, Korea, October 30-31, 2014

[7] B. W. Rhee, K.S. Ha, R.J. Park, J.H. Song, A Study of the Effect of Various Design Parameters on the Natural Circulation Flow Rate of the Ex-Vessel Core Catcher Cooling System of EUAPR1400, paper no 12133, Proceedings of ICAPP 2012,

[8] B. W. Rhee, K.S. Ha, R.J. Park, J.H. Song and Revankar, S.T, A scaling study of the natural circulation flow of the ex-vessel core catcher cooling system of EU-APR 1400 for designing a scale-down test facility for design verification, Proceedings of ICAPP 2012, June 24-26, 2012, Chicago, USA