CFD Simulation Concerning Effect of Aligned Obstacles on Downward Facing CHF

Yeon Soo Kim, Uiju Jeong, 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

Many safety features are provisioned to mitigate postulated severe accidents in nuclear power plant. An example is ex-vessel core catcher system, which is envisioned for European Advanced Power Reactor 1400 (EU-APR1400). It is installed especially to cope with direct containment heating (DCH), high pressure melt ejection (HPME), steam explosion, and molten corium concrete interaction (MCCI).

Currently, the thermal safety limit such as the CHF in cooling channel is being studied by some researchers. Korea Atomic Energy Research Institute (KAERI) has conducted many experiments to assess cooling performance of core catcher [1].

However, it is considered that the effect of studs should be investigated when obtaining the CHF data because the stud-induced eddies and corresponding turbulent motion were expected to influence the bubble behavior near the heater surface. Pioro et al. confirmed the adverse effect of flow obstacles at lower mass fluxes and lower qualities [2]. So single phase CFD simulation was performed to explain the observed CHF variation due to the existence of the studs in the cooling channel. It was assumed that the stud-induced eddies and corresponding turbulent motion in two-phase flow would be proportional to that in single-phase flow for simplicity. The assumption is considered reasonable because the liquid and gas regions are clearly stratified as boiling occurs at a downward facing flat surface. Kazuo et al. attempted to confirm the effect of two-phase flow through single phase CFD simulation [3].

Therefore in this study, the single phase 2D simulation results are presented and discussed to explain the effect of stud on the CHF variation with regard to the induced eddies and corresponding turbulent motion.

2. Core Catcher System

2.1 Ex-vessel Core Catcher System

Severe accidents can be progressed into in-vessel and ex-vessel events depending on vessel damage. Ex-vessel event refers to the condition, where the vessel fails. The ex-vessel core catcher is prepared for such event.

Fig.1 shows a conceptual design of ex-vessel core catcher system developed by KAERI. When the molten corium falls off to the core catcher body, the water stored in IRWST floods the core catcher by gravity and the decay heat from the corium is removed by both top flooding and boiling induced natural circulation of water in the cooling channel. The liquid-gas mixture in the channel flows from the inlet to the outlet, and then the mixture would be separated at the down-comer, which causes the natural circulation by density difference.

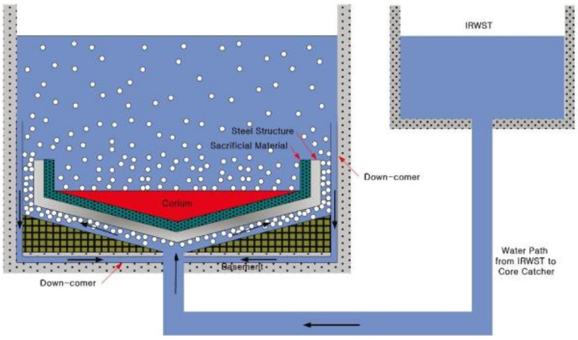

Fig. 1. Conceptual design of the core catcher system with natural circulation [4].

2.2 Cooling Channel in the Core Catcher System

The main features of cooling channels in core catcher system are as followings. First, it is tilted about 10 degrees and the bubble layer becomes clearly stratified by downward facing boiling heat transfer. Second, the size of core catcher system is so large, 6 m in width and 16 m in length, that it has large heat transfer area. Finally, there exist obstacles, called studs, which support massive core catcher body.

2.3 Role and Effect of Studs in the Cooling Channel

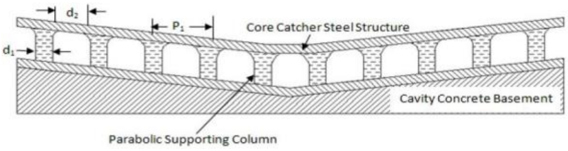

Fig. 2. Enlarged side view of the cooling channel [5].

Many studs play a supporting columnar structure for loading of core catcher body and therefore provide the cooling channel between core catcher steel structure and cavity concrete basement as shown in Fig. 2. But, it also acts as an obstacle to the flow. So water stagnates behind the stud and causes separated flow. The stagnant flow behind the stud would increase bubble residence time on the heat transfer surface, and thereby likely to reduce the CHF. On the other hand, the stud-induced eddies and corresponding turbulent motion would facilitate bubble detachment process and subsequently provide positive influence on the CHF. As mentioned above, the studs are double-edged.

3. Numerical Modeling

In this study, ANSYS FLUENT 17.0, a commercial CFD analysis tool was used.

3.1 Modeling of the Coolant Channel

In the modeling of the coolant channel, detached eddy simulation (DES) was used to capture the instantaneous turbulent motion, which cannot be captured by RANS based models. In fact, DES model is originally suitable for 3D simulation but it is executed by 2D simulation to reduce the calculation time. Thus, coolant channel was modeled in 2D and the size was set to 131.5 mm in width and 515 mm in length based on the experimental equipment in our laboratory.

3.2 Grid Generation

Only hexahedron grid was used to reduce the number of grids and to increase the quality at the same time. The number of grids was about 100,000. Because this study focuses on the effect of the studs in turbulent flow, the fine grid was generated only near the studs. For circular and elliptical studs, an O-type grid was used near the studs.

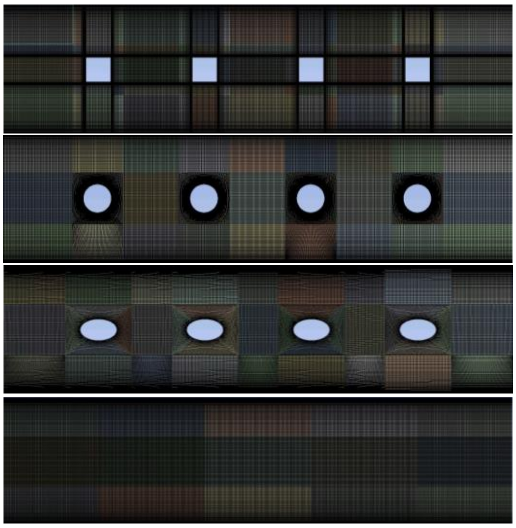

Fig. 3. The grids generated (a) rectangular stud (b) circular stud (c) elliptical stud and (d) bare stud

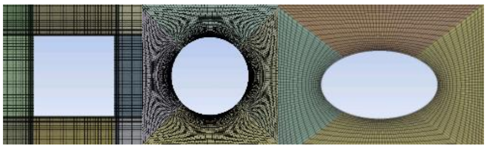

Fig. 4. The grids generated near (a) rectangular stud (b) circular stud (c) elliptical stud

3.3 Solver Settings

First, we simulated in steady state condition to improve convergence stability. After the continuity decreases near approximately 1×10^{-4}, the simulation was converted to transient to see how the vortex came up by the stud and changed over time. The density and viscosity of the water were determined at saturation temperature at atmospheric pressure. The stud surface and wall were given no-slip conditions.

Table I. FLUENT Solver Setting

	Inlet velocity :	0.2 m/s
	Model :	Delayed DES
	RANS Model :	Realizable k-ε
Spatial Discretization	Gradient :	Green-Gauss Cell based
	Pressure :	Second Order
	Momentum :	Bounded Central Differencing
	Turbulent kinetic energy :	First Order Upwind
	Turbulent dissipation rate :	Frist Order Upwind
	Pressure-velocity coupling scheme :	PISO
	Transient Formulation :	Bounded Second Order Implicit
Calculation	Time Stepping Method :	Fixed
	Time Step Size :	0.0001sec
	Number of Time Steps :	100000
	Max iterations / Time Step:	15

4. Comparison of Numerical Analysis and Experimental Result

There are many parameters indicating the existence or strength of turbulence such as turbulence kinetic energy (TKE) or turbulence intensity. But turbulence intensity can be estimated to some extent because of the definition itself but it requires more complex calculation than TKE. Thus TKE could be considered more useful. In addition, turbulence always accompanies eddy, whose strength and direction can be expressed in vorticity. Also because the CHF increases with the increasing mass flux, velocity could be an important factor. Therefore three parameters were selected for the analysis. They are TKE, vorticity, and velocity.

After running a single-phase CFD simulation, a transient 10-second animation about TKE, vorticity, and velocity were created to see behavior of the stud-induced eddies over time. When the representative figure was captured in region between the third and fourth stud, the probe function was used to obtain data near the eddy.

4.1 Turbulent Kinetic Energy

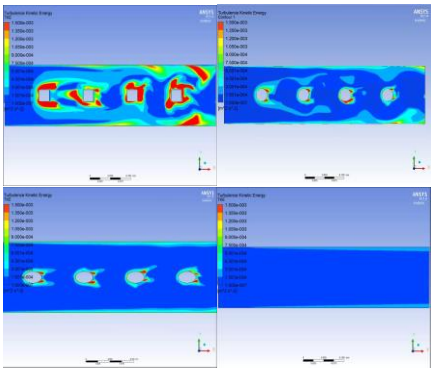

Fig. 5. Turbulence kinetic energy contours with variation of stud shape (a) square (b) circular (c) elliptical stud (d) bare

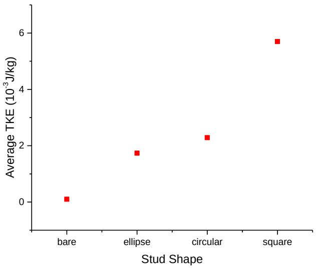

Fig. 6. A graph representing the stud induced turbulent eddy containing turbulence kinetic energy with variation of shape of stud

As seen in Figures 5 and 6, the highest TKE was seen in the square. It means that the stud-induced eddies and its energy was highest in the square studs.

Averaged value of square stud was 5.70×10^{-3} J/kg and that of circular stud was 2.29×10^{-3} J/kg, ellipse 1.74×10^{-3} J/kg, and bare 0.10×10^{-3} J/kg.

In the case of the square stud, TKE generated from the preceding stud was transferred intactly to the stud located right after the stud. In the case of circular and elliptical studs, however, considerable amount of the TKE was dissipated to surrounding flow before the eddies were reached to the studs located behind.

4.2 Vorticity

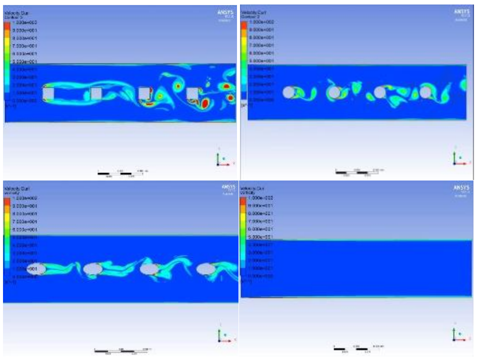

Fig. 7. Vorticity contours with variation of stud shape (a) square (b) circular (c) elliptical (d) bare

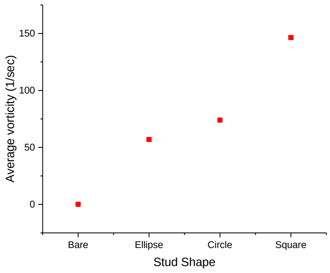

Fig. 8. A graph representing the stud induced turbulent eddy containing vorticity with variation of shape of stud

Figure 7 shows similar trend to that of TKE distribution. It shows that vorticity of the stud-induced eddies is proportional to its TKE, and the stud-induced turbulent motion takes form of vortex flow. As seen in Fig. 8, averaged value of the square case was the highest at 116.34 sec^{-1} and that of circular case was 62.37 sec^{-1}, ellipse 56.94 sec^{-1}, and bare 0.02 sec^{-1}.

4.3 Velocity

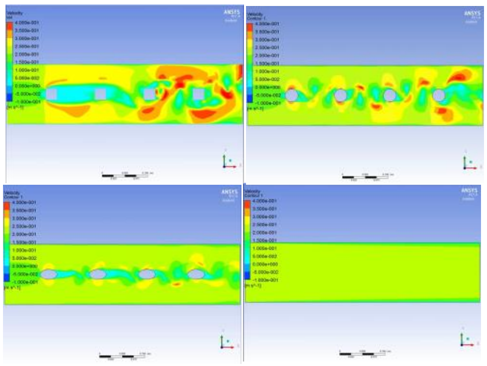

Fig. 9. Velocity contours with variation of stud shape (a) square (b) circular (c) elliptical (d) bare

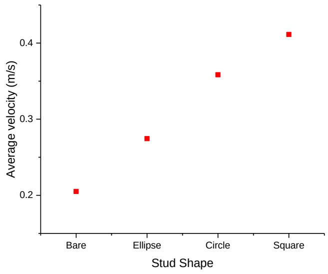

Fig. 10. A graph representing the stud induced local peak velocity with variation of shape of stud

Compared to TKE and vorticity, trend of the stud-induced local peak velocity with stud shape is most similar with that of experimental data on the critical heat flux, shown in Fig. 11. In the case of square, the local flow with high kinetic energy induced by the stud actively crosses the stagnant region between the studs, and the average peak velocity was 0.41 m/s, which is about two times the inlet velocity. In case of circular stud, the local flow with high kinetic energy also crosses the stagnant region but with narrow region and less peak velocity of 0.36 m/s, but larger than that of the elliptical stud case of 0.27 m/s.

However, in case of ellipse, there was less acceleration caused by the stud compared to other stud cases. Also, the cross flow observed in square and circular cases didn't appear. The absence of the cross flow induced by studs in ellipse case would influence the negative effect on the turbulent motion and bubble detachment process at the region between studs.

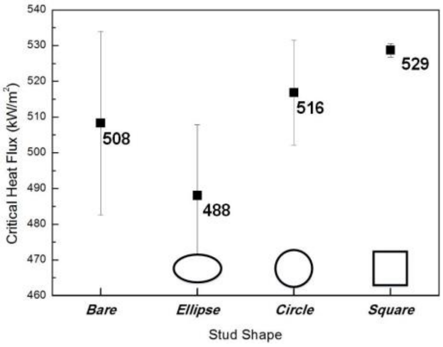

Fig. 11. CHF data by shape of stud

As mentioned in Section 2, studs have a positive effect in inducing turbulence and bubble detachment and an adverse effect of confining the bubble in the region between studs.

In cases of square and circular stud, values of TKE, vorticity, and local peak velocity were measured relatively higher than the others. It implies that the stud-induced eddy and corresponding turbulent motion were significantly strong in the stagnant region. So enhanced bubble detachment effect caused by the stud could be expected and this was confirmed by the experimental results as seen in Fig.11. The highest CHF occurred in especially square and circle in order. This suggests that the presence of stud contributes to the CHF enhancement.

However, the ellipse has the lowest CHF value than the bare despite the presence of stud. It can be explained that the streamlined shape of ellipse reduces the stud-induced eddies, corresponding turbulent motion, and the cross flow over the stagnant region with high kinetic energy. This suggests that the adverse effect of confining the bubble was more dominant.

5. Conclusion

In this study, single-phase simulations using DES model were performed to explain the CHF variation due to the existence of the studs in the cooling channel by capturing instantaneous stud induced flow structure including turbulence. Three types of stud shapes were selected: square, circle, and ellipse. The conclusions from this study can be summarized as follows.

- In the presence of studs, the value of TKE, vorticity, and local peak velocity were significantly higher than those in bare case. It implies that the stud-induced eddy and corresponding turbulent motion were active. Thus significant bubble detachment effect caused by the stud could be expected. This was confirmed by the experimental results. This suggests that the presence of stud contributes to the CHF enhancement.
- Lower CHF value was measured in case of the ellipse compared to that of bare. This can be explained by considering that the streamlined shape of ellipse structure reduces vortex flow, turbulent motion, and the cross flow significantly, while confinement effect by columnar structure stands out as importance in CHF triggering mechanism. This suggests that the presence of the streamlined stud contributes to the CHF reduction.
- The experimental results show that the CHF enhancement effect was the most dominant in case of the square or circular studs. Above all, the square stud can be considered the most optimized shape considering the safety margin and manufacturing cost.

The simulation in this study was conducted with the assumption. However, the actual phenomena are under two-phase flow condition. Thus for improved results, more experiments will be conducted and compared with the CFD simulation at low heat flux.

REFERENCES

[1] Ha, Kwang Soon, Park, Rae Joon, Wi, Kyung Jin, & Thanh, Thuy Nguyen Thi (2014). KAERI Activities on the Cooling Performance of Ex-vessel Core Catcher. *Proceedings of the KNS 2014 Fall Meeting,* (pp. 1CD-ROM). Korea, Republic of: KNS.

[2] Pioro, I. L., Groeneveld, D. C., Doerffer, S. S., Guo, Y., Cheng, S. C., & Vasić, A. (2002). Effects of flow obstacles on the critical heat flux in a vertical tube cooled with upward flow of R-134a. International Journal of Heat and Mass Transfer, 45(22), 4417-4433.

[3] Ikeda, K., Makino, Y., & Hoshi, M. (2006). Single-phase CFD applicability for estimating fluid hot-spot locations in a 5× 5 fuel rod bundle. Nuclear Engineering and Design, 236(11), 1149-1154.

[4] 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.

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