CFD Analysis of Subcooled Boiling Phenomena in the crud DeposItion Simulator for Nuclear energY (DISNY)

Hyeon Ji Kim, Ji Yong Kim, In Cheol Bang*
Department of Nuclear Engineering
Ulsan National Institute of Science and Technology (UNIST).,
50 UNIST-gil, Ulju-gun, Ulsan, 44919, Republic of Korea
**Corresponding author: icbang@unist.ac.kr*

1. Introduction

The crud DeposItion Simulator for Nuclear energY (DISNY) is an experimental loop for crud deposition to evaluate the effective thermal properties of the crud [1]. The crud could increase the thermal resistance of fuel rods during normal and transient operations. Still, a database of crud thermal properties is lacking in the actual nuclear power plant operating environment. Therefore, the DISNY loop was designed to simulate the normal and transient operating conditions of the Korean pressurized water reactor (PWR). The test section of DISNY in Fig. 1 has an annular flow path simulating a single subchannel of the PWR. The working fluid introduced from the inlet boundary goes through the downcomer and rises up along the chimney channel. Passing the entrance of the chimney channel, the working fluid exchanges heat with a zirlo heater that simulates nuclear fuel. At this time, the working fluid flow should be fully developed to simulate a PWR subchannel where the crud is formed. Insufficient entrance length improves heat transfer, which may underestimate the effect of crud on fuel rods.

The wall of the zirlo heater is superheated like a real fuel rod in a crud-forming subchannel, while the temperature of the working fluid is below the saturation point, resulting in subcooled boiling. At this time, the bubble departure-induced convection could enhance the local heat transfer between the zirlo heater and the working fluid. An analysis of the local HTC enhanced by the boiling phenomena should be preceded in order to accurately evaluate the effective thermal properties of the crud.

In this study, a computational fluid dynamics (CFD) analysis of the test section of the DISNY was performed using ANSYS Fluent 19.0 [2]. The velocity distribution of the working fluid showed that the entrance length is sufficient for working fluid flow to be fully developed and the DISNY test section can simulate the top 0.3 m PWR single subchannel. With the multiphase model, the local HTC, considering the effect of subcooled induced convection was evaluated, which should be considered in evaluating the effect of crud on a fuel rod through experiments using the DISNY loop.

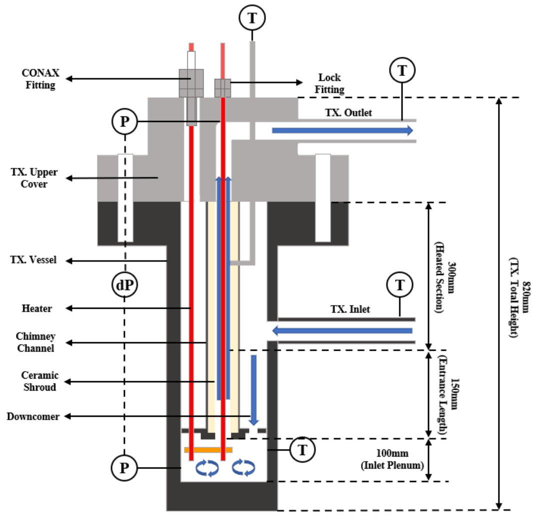

Fig. 1 Cross-sectional view of the DISNY test section

2. Numerical methods

The computational model of the DISNY test section in Fig. 2 is divided in two sections: the fluid flow domain and the solid heater domain. Solid section excluding heater were not taken into account, all geometric properties were implemented in fluid region. The working fluid introduced from inlet boundary descends along the downcomer, passes through the chimney channel, and exits to the outlet boundary.

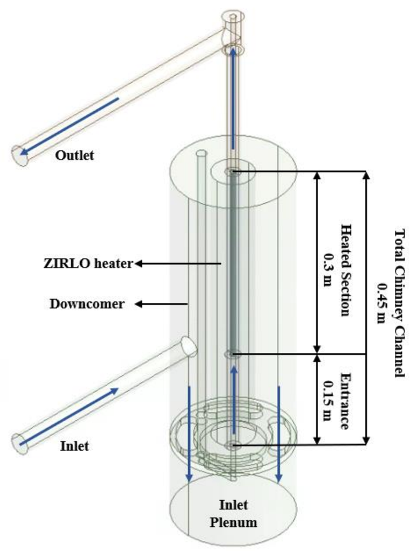

Fig. 2 CFD computational model of the DISNY test section

2.1 Analysis Method

Flow inside test section is two-phase by subcooled boiling. The eulerian multiphase model was used to solve mass, momentum, and energy conservation of each phase. The correlation used to model the interfacial momentum and energy transfer are listed in Table I. The shear stress transfer (SST) k-ω model was used to model turbulence for each phase, treating water as continuous while the vapor as dispersed. Wall boiling phenomena was modeled based on RPI wall boiling model which defines the total heat flux as partitioned into convective, quenching and the evaporative heat fluxes [3]. The boiling model parameters for bubble departure diameter, frequency of departure and nucleation site density were also listed in Table I.

Table I: Models used in the simulation

Physics	Model
Drag force	Ishii
Lift force	Moraga
Wall lubrication	Hosokawa
Turbulent Dispersion	Burns-et-al
Turbulence Interaction	Sato
Interfacial Area	symmetric
Bubble departure diameter	Tolubinski - Kostanchuk
Frequency of bubble departure	Cole
Nucleation site density	Lemmert - Chawla

2.2 Computational Grid

The mesh of the computational model in Fig. 3 and Fig. 4 was written by ANSYS Meshing with the hexahedral and tetrahedral shapes. Prism meshes were stacked on the wall of the chimney channel, the section where boiling could occur. The number of 2,157,600 nodes and 2,082,000 elements for the chimney channel and the total number of 3,701,328 nodes and 4,683,388 elements were used.

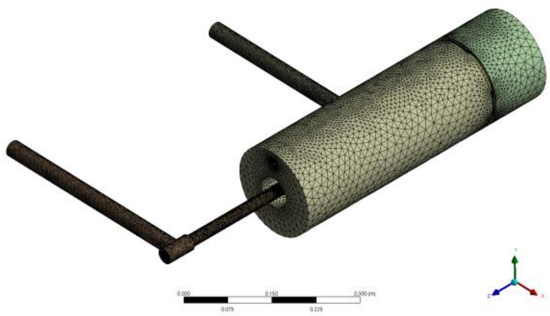

Fig. 3 Mesh of the whole DISNY test section

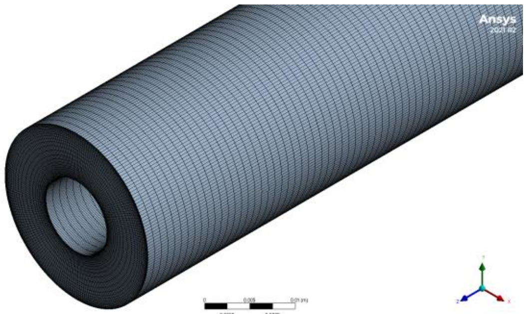

Fig. 4 Mesh of the chimney channel in the DISNY test section

2.3 Boundary Conditions

Table II lists the boundary conditions of CFD analysis selected based on the test matrix. The volumetric heat source of crud deposition condition was applied to zirlo heater section simulating cladding of fuel rod. PWR minimum designed flow rate and coolant temperature at the top 0.3 m at the high-temperature flow path, and pressure of PWR normal operating conditions were applied.

Table II: Boundary conditions

Boundary Location	Boundary type	Value
ZIRLO heater	Volumetric heat source	364.8 MW/m^3
Inlet	Mass-flow inlet	1.055 kg/s
	Temperature	613.15 K
	Pressure	155 bar
Outlet	Pressure-out	-

3. Results and Discussion

3.1 Flow stabilization

The area to be simulated by the DISNY is the region top 0.3 m of the fuel rod assembly where the flow of coolant is fully developed. Therefore, the entrance length within the designed test section should be evaluated whether it is long enough. Fig. 5 shows the velocity distribution of working fluid at the chimney channel in the boundary conditions of Table 2 (L = 0 at the entrance of the chimney channel). The velocity distribution tends to become stable depending on the fluid flow direction. The zirlo heater is positioned at L/D = 6.73 ~ 20.18, and the flow is close to fully developed from L/D = 10. Thus, the entrance length could be seen as properly designed.

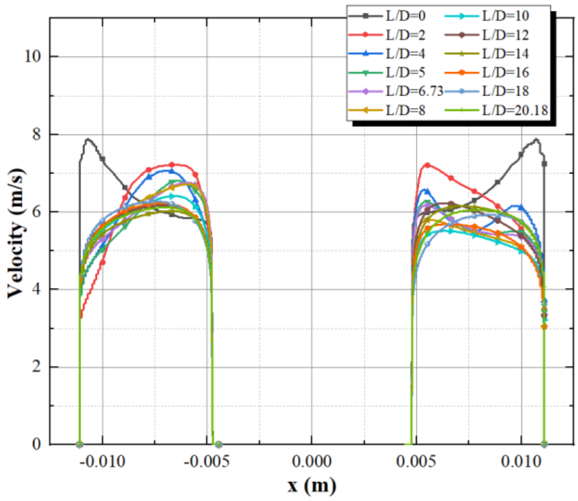

Fig. 5 Velocity distribution of the working fluid per Length / Diameter ratio

3.2 Local Heat transfer coefficient

Fig. 6 shows the heater wall temperature and the bulk coolant average temperature in the heated section of Fig. 2. The heater wall temperature is relatively low at the entrance and shows that heat transfer is improved due to turbulence generated by a rapidly reduced flow path. As the flow is stabilized, the wall temperature rises but decreases locally in the subcooled boiling section. It is considered the effect of bubble-induced convection by boiling phenomenon. Therefore, as shown in Fig. 7, HTC increases in the boiling section. HTC was obtained by Newton's law of cooling based on temperature in Fig. 6. The effects of flow characteristics of experimental equipment and subcooled boiling phenomena on HTC should be considered when evaluating the thermal properties of crud through experiments.

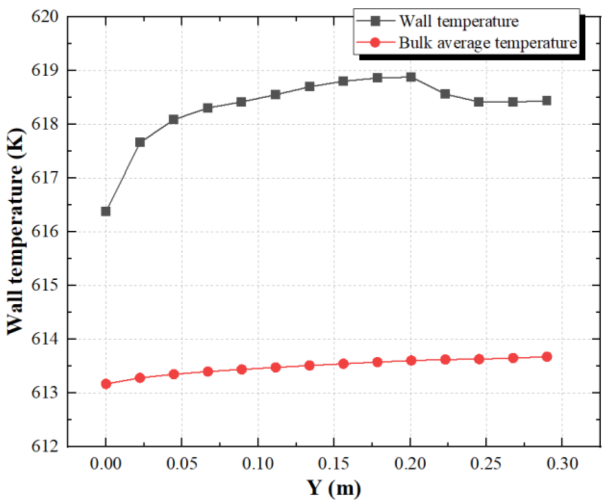

Fig. 6 Temperature distribution of the heater wall and the average bulk coolant

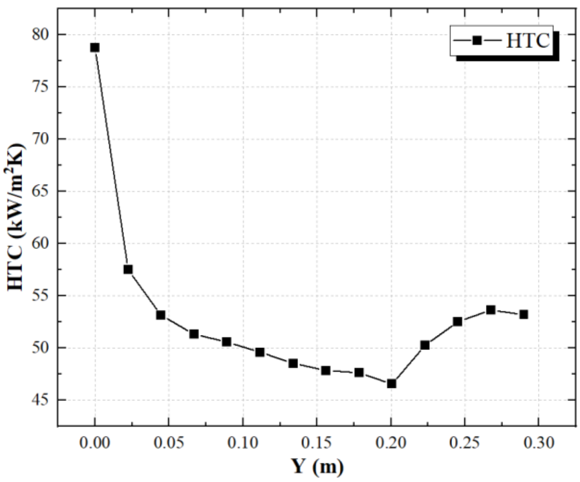

Fig. 7 Heat transfer coefficient between the heater wall and the coolant

4. Conclusion

In flow boiling condition, the working fluid velocity distribution per height and local heat transfer coefficient of the chimney channel of the DISNY test section were analyzed using ANSYS Fluent 19.0. Due to abrupt change in the flow field, the velocity distribution shows incompletely developed flow at the entrance of the chimney channel. The flow stabilized as the working fluid proceeds, and fully developed at about L/D = 10. The entrance length was appropriately selected, in that, a zirlo heater presences at L/D = 6.73 ~ 20.18. Thus, the DISNY test section can simulate a top 0.3 m subchannel of fuel assembly. The HTC locally increased at the entrance and the subcooled boiling region. Strong turbulence due to changes in the flow field and convection due to bubble rise increased the local HTC. The HTC should be corrected based on the analysis results when evaluating crud thermal properties, to prevent underestimating crud effect on a fuel rod.

ACKNOWLEDGMENTS

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety (KoFONS) using the financial resource granted by the Nuclear Safety and Security Commission (NSSC) of the Republic of Korea. (No. 2106022)

REFERENCES

[1] J.Y. Kim, H.J. Kim, Design Study of CRUD Thermal Properties Characterization Facility DISNY under Pressurized Water Reactor Normal Operating Condition, Advances in Thermal Hydraulics (ATH-2022), June 12-16, 2022, Anaheim, CA

[2] ANSYS Fluent Theory Guide, ANSYS Inc., Canonsburg, 2013

[3] N. Kurul, M.A. Podowski, On the modeling of multi-dimensional effects in boiling channels, ANS Proceedings of 27^{th} National Heat Transfer Conference, 1991, Minneapolis, MN