Conjugate Heat Transfer Analysis of the High Temperature Heater for Supplying Superheated Steam to a Lab-scale HTE Device

Churl Yoon*, Sung-Deok Hong, Hong Sik Lim, Chan Soo Kim
Korea Atomic Energy Research Institute, Yuseong-Gu, Daejeon, Korea, 305-600
**E-mail: cyoon@kaeri.re.kr*

1. Introduction

A Lab-scale helium loop for simulating a VHTR (Very High Temperature Gas Cooled Reactor) is now under constructing at the Korea Atomic Energy Research Institute. The Lab-scale helium loop will be connected to 30kW capacity "High Temperature Electrolysis (HTE) system" as a function of high-temperature steam supply to the HTE device (Figure 1). A high-temperature heater (HTH) heating the 3.0MPa helium gas up to 1000°C is one of key components in the Lab-scale helium loop. The HTH uses a non-metal heating element which can withstand temperatures in excess of 2000°C in an oxygen and moisture free environment. [1]

Hong, et al.[1] defined the design requirements and proposed a design output for a high-temperature heater that has the following operating conditions:

Power,	55 kW
Design pressure,	6.0 MPa
Outlet temperature,	1000 °C
Flow rate (helium, max.),	1.0 kg/min

In their study [1], the proposed design was validated by using the GAMMA+ thermal-hydraulic system code. The purpose of this study is to develop a CFD (Computational Fluid Dynamics) model for analyzing conjugate heat transfer phenomena inside the HTH, and to validate the proposed design with detailed-level simuation.

2. High Temperature Helium Heater

Fig. 1 shows a schematic of integrated 30kW HTE system. Pressurized helium flow is maintained by circulator. Helium is heated from room-temperature to 950 ℃ by electric power. Purified water is pressurized by pump. It is evaporated by helium-water evaporator. The steam is super-heated by helium-steam-air heat exchanger. Air is pre-heated by electric power and heated to 820 ℃ by helium-steam-air heat exchanger. It is supplied to the anode of HTE stack. The product from the cathode is mixture of hydrogen and steam. It is cooled, condensed and separated. [2]

Helium heating system in the lab-scale helium loop of the Figure 1 is composed of two heaters; preheater and main heater (HTH). The preheater heats helium gas up to 500°C and sends it to the inlet of the main heater. The main heater is a high temperature heater (HTH) which heats the 500°C helium gas up to 1000°C.

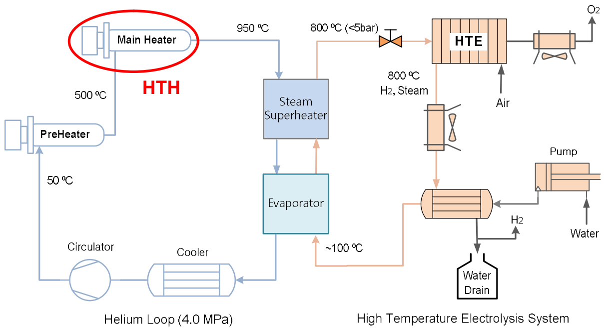
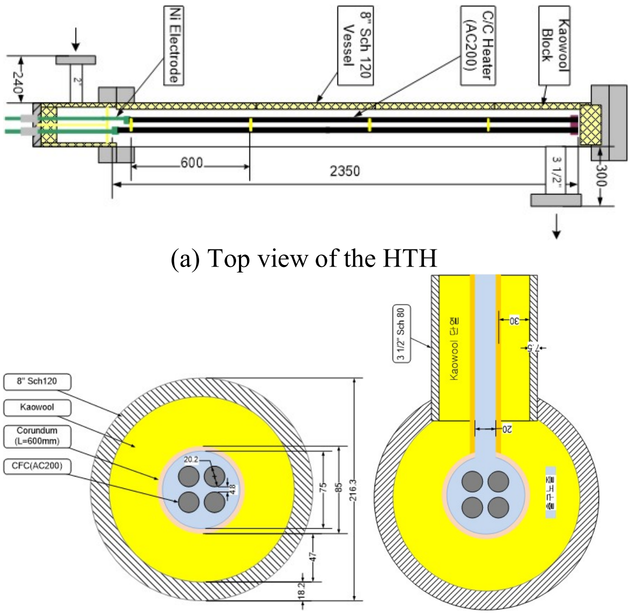


Fig. 1. Layout of a helium loop connected to HTE system



(a) Top view of the HTH

(b) Cross-section of the Heater (c) Exit cut-view

Fig. 2. High-temperature heater for the HTE system

Figure 2 shows the horizontal and cross-sectional views of the internal of HTH, which was installed horizontally. The stainless-steel vessel of the HTH heater is internally insulated by Kaowool ceramic fiber to protect the vessel from the 1000°C helium. A Corundum liner provides a robust separation between the insulator and the flow channel. The material of heater array is a Carbon Fiber Composite (CFC) that can withstand over 2000°C in the oxygen and moisture free environment. The heater array is electrically insulated from each other by Boron-Nitride (BN) ceramic spacers of which are equally spaced horizontally 600mm apart.

3. Analysis Model and Results

A CFD model has been constructed and used to perform preliminary computations of flow in the HTH. The commercial CFD code, ANSYS CFX Ver. 2021R2 [3], was used for these simulations.

3.1. CFD Model

Since the long heater vessel was horizontally installed, the whole fluid flow and heat transfer regions from the starting point of CFC heaters up to the exit of the outlet pipe were set as the computational domain to consider buoyancy effect. Inside the helium flow passage, heat from the 4 CFC heaters transferred to surrounding fluid and solid structures by convection and radiation as well as conduction. Reynolds numbers of the given operating conditions for the Case-1 in Hong, et al.[1] are ranged from 533 to 8,360, and the internal fluid flow would be very complicated near the spacers and the corner near the entrance to the exit pipe. The k-ω based SST (shear stress transport) turbulence model with automatic wall function and the P1 radiation model were adopted for turbulent flow and radiative heat transfer predictions. The k-ω based SST turbulence model is good in predicting the onset and amount of flow separation from smooth surfaces. The P1 radiation model solves the moment equations of radiative intensity being assumed isotropic, which gives reasonable accuracy without too much computational effort.[3]

Figure 3 shows the full computational domain and meshes near the inlet of fluid region. With 6 prizm layers in the near-wall 3.0 mm thickness to assure y^+ values less than 2.0 in most fluid regions, the total number of nodes and elements are 38,578,263 and 25,073,597.

In this preliminary calculation, following assumptions and simplifications are established:

- Inlet velocities were assumed uniform with a given mass flow rates (MFR).
- Solid surfaces at the inlet, the exit, and the outer end surface of the main vessel were set as adiabatic boundary conditions.
- Inlet fluid temperatures were assumed as 500°C.
- End mount was assumed to be located 2.0cm away from the end insulation plug.
- Volumetric heat generation rates of the 4 CFC heaters were set to be uniform.
- Heat flux BC at vessel outer surface was set as the following equation (1) [4], with the environmental temperature (T_∞) of 30°C.

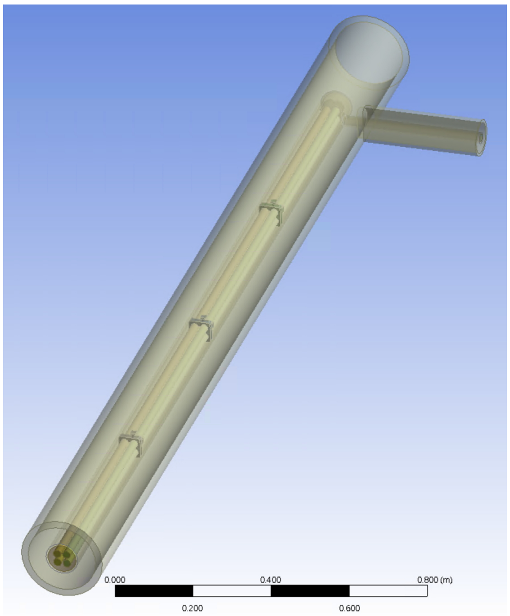
$$q''_w = h\left(T_w - T_\infty\right) + F\varepsilon\sigma\left(T_w^4 - T_\infty^4\right) \quad (1)$$

where, h is convective heat transfer coefficient simply set to be a constant value of 10W/m^{2}K, F is a view factor, ε is emissivity, and σ is Stefan-Boltzmann constant.

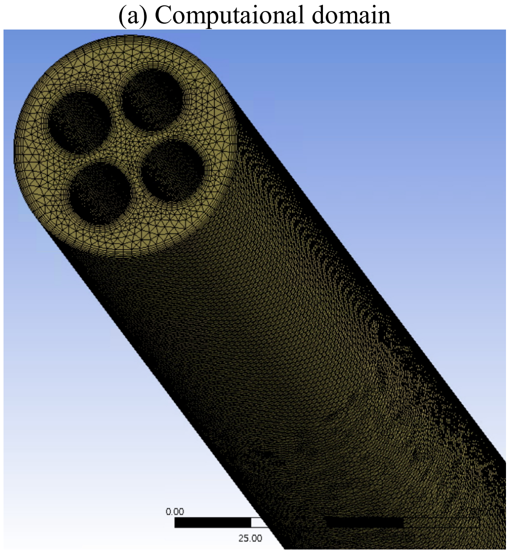
- Emissivity valus of all the electric heaters and structural surfaces were assumed to be 0.9. Only the emissivity of the insulation-Helium inteface surface was exceptionally set to be 0.5.
- The effective conductivity of Kaowool ceramic fiber insulation was expressed as

$$\lambda_{eff} = 0.0201 + 6.04\times10^{-4}\,T(K).\ [\text{W/m/K}] \quad (2)$$

For the more precise prediction, the above assumptions and simplifications could be improved after comparison with measured experimental data.



(a) Computaional domain



(b) Meshes near the fluid inlet

Fig. 3. Computaional doamin and meshes for the HTH analysis

3.2. Results and Discussion

For comparison with the GAMMA+ results[1], the same five combinations of the gas mass flow rate and heat generation rate were simulated for helium gas flow. Table 1 give the comparison of the calculation results by using CFX CFD tool with the GAMMA+ calculation results. Note that the maximum vessel temperatures and the heat losses through the vessel outer surface in the CFD analyses were extracted only at the main vessel part, excluding the smaller outlet pipe. Compared to GAMMA+ results, the CFD analyses give lower temperatures in predicting the maximum temperatures of the heaters and the vessel, and more heat losses through the vessel outer surface. The reason for this is thought to be caused by assuming the high heat flux boundary condition at the vessel outer surface including radiation heat transfer by equation (1).

Figure 4 presents the temperature contour on some selected cross-sections for the case of o.6kg/min MFR and 29.6kW heat load. The five cross-sections are located 0.0m, 1.0m, 1.5m, 2.2825m, and 2.625m away from the inlet, respectively. Due to the direct heating by radiation, the inner surface temperature of the insulator is higher than that of the He flows. The hot He gas area in the upper region of the coolant passage, caused by buoyant flows, was not quite apparent in figure 4, since radiative heat transfer is stronger than convective heat transfer in this case. However, the buoyant He flows induced by heating were observed in other detailed views.

Figure 5 shows the streamlines around the third spacer and the end mount for the case of 0.2kg/min MFR and 12.4kW heat load. Flow velocities were increased as passing through the obstacles (spacers) and going into the narrow outlet pipe. Since the end mount was installed 2.0cm away from the internal end wall, the fluid streamlines went around the end mount.

Table 1. Comparison of the CFD and GAMMA+ resultsfor the given design (Case-1) with helium gas flow.

MFR [kg/m]	Re (outlet)	Power [kW]	T_{max} (heater) [°C]	T_{max} (vessel) [°C]	Heat Loss [kW]
ANSYS CFX					
0.2	533	12.4	976.1	239.0	4.41
0.4	1070	21.0	1026.4	250.6	4.60
0.6	1580	29.6	1059.3	255.8	4.74
0.8	2100	38.5	1090.1	259.6	4.85
1.0	2310	47.2	1113.9	261.5	4.92
GAMMA+					
0.2	533	12.4	1084.9	338.5	3.76
0.4	1070	21.0	1068.7	335.3	3.59
0.6	1580	29.6	1105.9	340.9	3.68
0.8	2100	38.5	1194.5	360.0	3.94
1.0	2310	47.2	1269.4	372.9	4.23

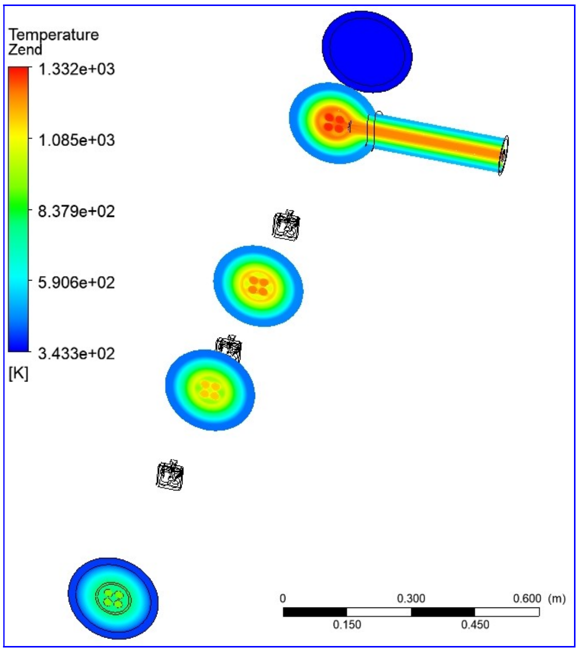
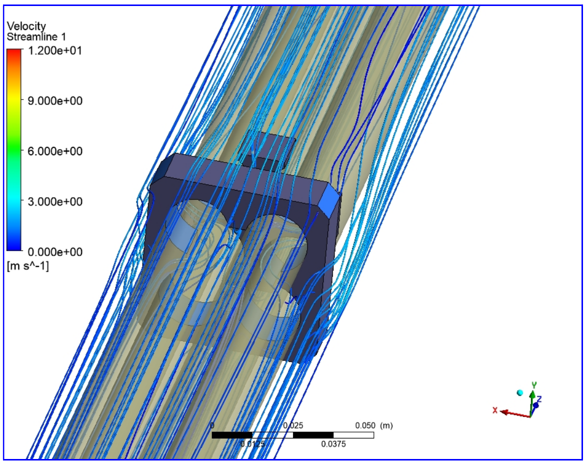
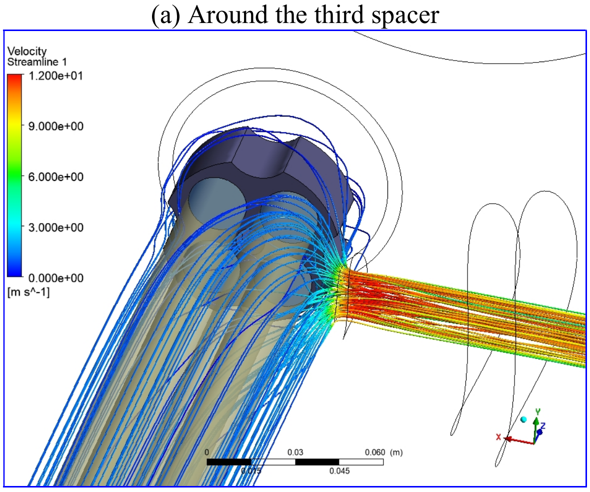


Fig. 4. Temperature contour on some selected cross-sections for the helium flow with o.6kg/min MFR and 29.6kW heat load



(a) Around the third spacer



(b) Around the End Mount

Fig. 5. Streamlines for the case of 0.2kg/min MFR and 12.4kW heat load

Figure 6 presents the temperature contours on the vessel outer surface for the five combinations of MFR and heat load. All the contours of (a) ~ (e) in figure 6 were drawn with the same contour level. Since the higher MFR is combined with the higher heat load, the case (e) shows highest maximum temperature on the vessel outer surface which were found near the outlet pipe entrance.

As an alternative coolant fluid, nitrogen gas was tested for HTH analysis. Table 2 summarizes the comparison between helium and nitrogen coolants. The nitrogen gas density at 3.0MPa and 750°C is 9.77890kg/m^3 while the helium gas density at the same condition is 1.40688kg/m^3. Figure 7 shows the temperature contours on some selected cross-sections for the hitrogen gas flow with 2.8kg/min MFR and 29.6kW heat load.

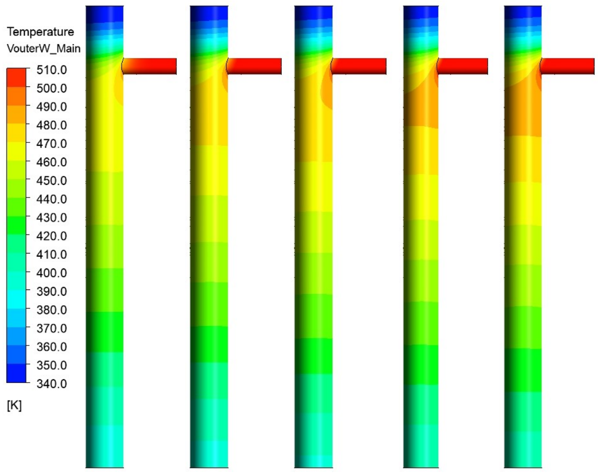
4. Conclusions

In this study, a CFD (Computational Fluid Dynamics) model for analyzing the conjugate heat transfer phenomena inside the HTH has been developed through the procedures of geometry setting, mesh generation, physical model setting, and computation. The CFD calculation gives quite reasonable results, comparing with the GAMMA+ results [1]. This CFD simulation methodology will be utilized for validating the proposed HTH design and provide insight of the conjugate heat transfer phenomena inside the HTH.

In these prelimanary calculations, any examination about various turbulence and radiation heat transfer models has not been performed. Also, the buoyancy turbulence was not considered. The *k-ω* based SST turbulence model and the P1 radiation model were adopted since they are robust and cost-efficient. In the case that any needs for design improvement or CFD model elaboration arise, additional investigation on the proper turbulence model as well as the radiative heat transfer model will be conducted in the future study.

Table 2. T/H differences between helium and nitrogen: comparing the CFD and GAMMA+ results (Case-1 with pressure = 3.0MPa and inlet temperature = 500°C)

Parameter	ANSYS CFX		GAMMA+	
	He	N_2	He	N_2
T_{max} (heater) [°C]	1059.3	1138.2	1106	1166
T_{max} (vessel) [°C]	255.8	259.2	341	302
Heat loss [kW]	4.74	5.96	3.68	2.83
Flowrate [kg/min]	0.6	2.8	0.6	2.8
Reynolds No.	1580	8260	1580	8260
Power [kW]	29.6	29.6	29.6	29.6



(a) (b) (c) (d) (e)

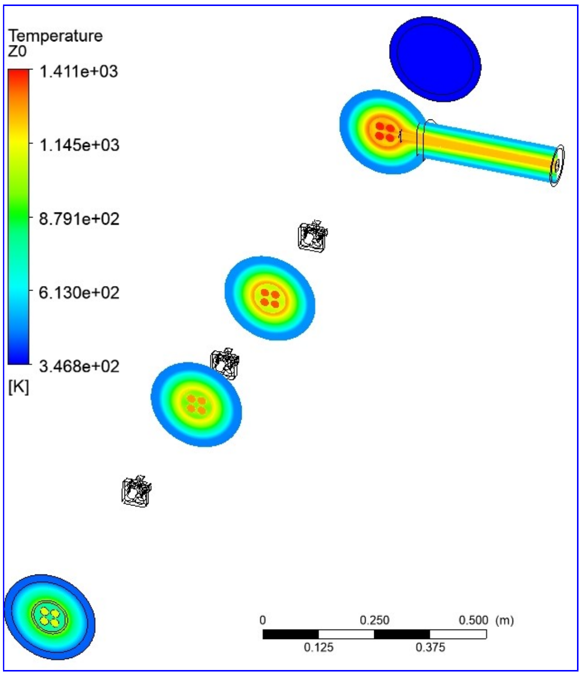
Fig. 6. Temperature contours on the vessel outer surface for cases of (a) 0.2, (b) 0.4, (c) 0.6, (d) 0.8, and (e) 1.0 kg/min MFR

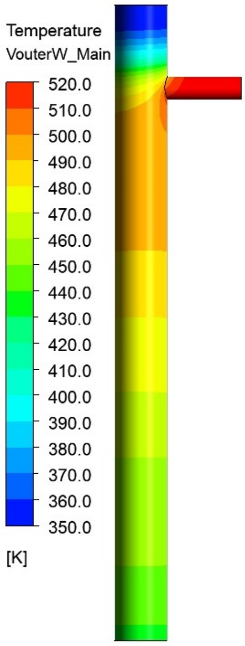
ACKNOWLEDGEMENTS

This work is supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (NRF-2021M2D4A2046777)

REFERENCES

[1] S. D. Hong, H. S. Lim, C. S. Kim, "High Temperature Helium Heater for 800°C Steam Supply to a Lab-scale HTE Device," KNS Autumn meeting, Changwon, 2021.
[2] B. H. Park, H. S. Kim, S.-D. Hong, C. S. Kim," A 30 kW High Temperature Electrolysis Experimental Facility for Nuclear Hydrogen Production," KNS Spring meeting, Jeju, 2022.
[3] ANSYS Web site, https://www.ansys.com
[4] C. Yoon, et al., "A CFD Analysis for Designing the VHTR Simulated Experimental Loop," KSME journal B, Vol. 34, No. 5, pp. 554~561, 2010. (in Korean)
[5] G. Melese and R. Katz, "Thermal and Flow Design of Helium-Cooled Reactors," American Nuclear Society, 1984.





(a) On selected cross-sections (b) On vessel outer surface

Fig. 7. Temperature contour on some selected cross-sections and vessel outer surface for the nitrogen gas flow with 2.8kg/min MFR and 29.6kW heat load