Transactions of the Korean Nuclear Society Spring Meeting, May 13-14, 2021

Applicability of Non-condensable Gas Model for S-CO_2 Ingress to High Pressure Water

Jae Jun Lee, Jeong Ik Lee*
Department of Nuclear and Quantum Engineering, KAIST
*Corresponding author: jeongiklee@kaist.ac.kr

KAIST Nuclear & Quantum Engineering

Introduction

- A Small Modular Reactor (SMR) is considered as a promising future reactor technology. For SMR development, many innovative power cycles have been proposed and among them, a Supercritical CO_2 (S-CO_2) power cycle is seriously being considered. A combination of mature water cooled reactor technology and an S-CO_2 power cycle is also one of the potential candidates.

- Generally, the conventional operating pressure of S-CO_2 power cycle is higher than the operating pressure of water cooled reactor. For this reason, if the intermediate heat exchanger fails, CO_2 would ingress to the waterside. This leads to an increase in system pressure, and it raises a question from a safety point of view.

- Thus, safety analysis code should have a capability to simulate the phenomenon. The authors tried to use the non-condensable(NC) gas model to analyze it. In short, the applicability of NC gas model in an 1-D thermal hydraulic (TH) analysis code is presented for the analysis of S-CO_2 ingress to high pressure water by comparing the experiment and the numerical simulation.

Non-condensable gas model

- Major parts of NC gas model those are adopted in the in-house code are introduced. The basic assumptions of the code are as follows.

1) The dissolved gas component is in mechanical equilibrium with the liquid phase
2) Diffusion in the liquid and wall heat transfer are neglected.
3) Interfacial heat transfer between gas and liquid phase is neglected. Only the energy transfer accompanied by mass transfer is considered.

- The in-house code solves of 3 continuity, 2 momentum and 2 energy equations. The below continuity equations correspond to the non-condensable gas, dissolved gas, and to liquid. The dissolution and release of NC gas are modeled as a mass transfer term (Γ).

Continuity equation set with non-condensable gas

$$\frac{\partial(\alpha_g\rho_g)}{\partial t}+\frac{1}{A}\frac{\partial(\alpha_g\rho_g V_g A)}{\partial x}=\Gamma_i$$

$$\frac{\partial((1-\alpha_g)\rho_l X_{n,l})}{\partial t}+\frac{1}{A}\frac{\partial((1-\alpha_g)\rho_l X_{n,l} V_l A)}{\partial x}=-\Gamma_i$$

$$\frac{\partial((1-\alpha_g)\rho_l (1-X_{n,l}))}{\partial t}+\frac{1}{A}\frac{\partial((1-\alpha_g)\rho_l (1-X_{n,l}) V_l A)}{\partial x}=0$$

where

$$\Gamma_i=\frac{(1-\alpha_g)\rho_l(X_{n,l}-X_{n,l,eq})}{\tau}$$

$$X_{n,l}=\frac{M_{Dissolved\ gas}}{M_{Dissolved\ gas}+M_{liquid}}$$

$$\rho_l=\frac{1}{v_l}=\frac{1}{v_{apparent}(X_{nl})+v_{liquid}(1-X_{nl})}$$

$$g: gas\ phase, l: liquid\ phase$$
$$n,l: dissolved\ gas\ in\ liquid, eq: equilibrium$$

For other equations and numerical techniques, MARS code is referred

Constitutive equations for non-condensable gas

- For interfacial friction between the gas and liquid, drag coefficient method is adopted in the in-house code. Ishii and Zuber proposed bubble drag coefficient for various regions.

$$Undistorted\ particle\ regime \quad C_{D,1}=\frac{24}{Re_b}\left(1+0.1Re_b^{0.75}\right)$$

$$Distorted\ particle\ regime \quad C_{D,2}\cong\frac{2}{3}D_b\sqrt{\frac{g\Delta\rho}{\sigma}}$$

$$Churn\ turbulent\ flow\ regime \quad C_{D,3}=\frac{8}{3}(1-\alpha_g)^2$$

$$C_D=\max\left(C_{D,1},min\left(C_{D,2},C_{D,3}\right)\right)$$

- Pure fluid properties are calculated using REFPROP program. Apparent volume of CO_2 is calculated using the model proposed by Hu, Q et al. Duan and Sun solubility prediction model is used.

Simulation of CO_2 injection into water

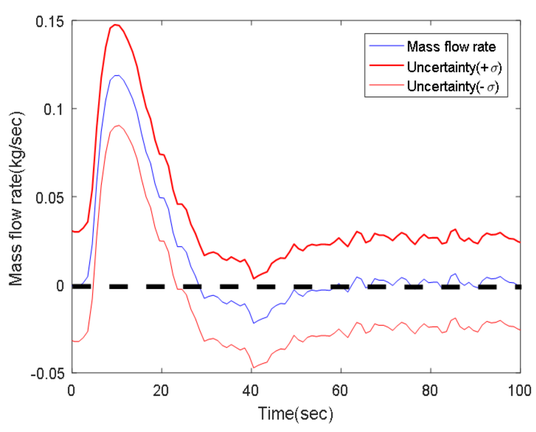
Description of experiment and simulation

- The nodalization and the experimental facility are presented. As shown in the schematic, the left tank is filled with CO_2 and CO_2 flows into the right tank filled with water from below. In the simulation, only the right tank is modelled and CO_2 injection is dealt as the boundary condition.

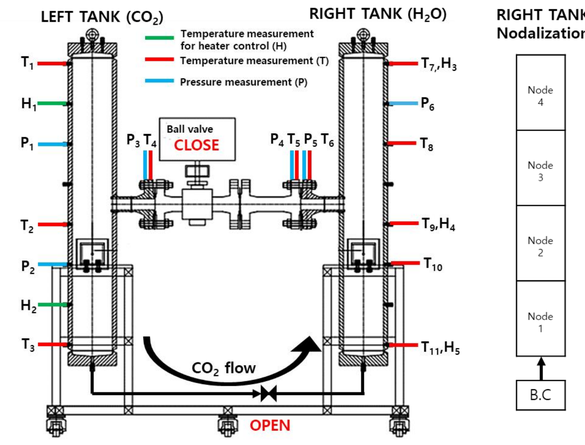
- The mass flow rate of CO_2 was calculated with measured pressure and temperature data in the experiment.

$$\dot{m}=\frac{V_{tank}(\rho_{t+\Delta t}(P,T)-\rho_t(P,T))}{\Delta t}$$

$$\Delta t: measurement\ time\ interval$$



✓ Mass flow rate of CO_2



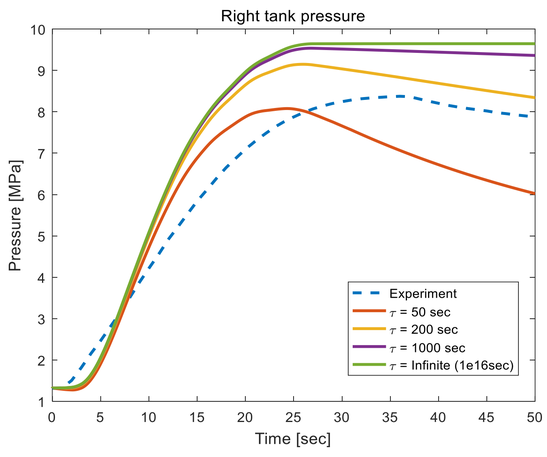
✓ Schematic of the experimental facility and nodalization for simulation

Simulation results

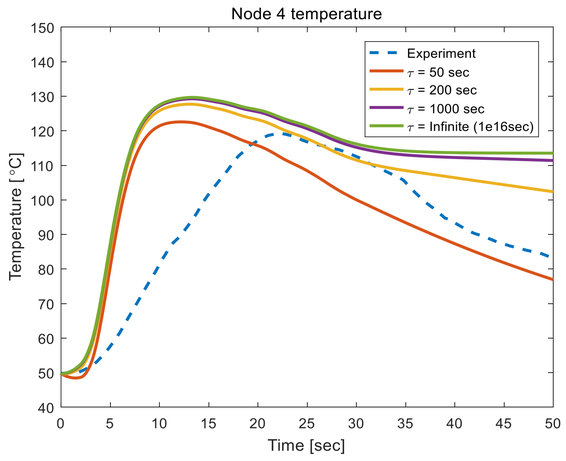
- To identify the effect of the time constant, simulation results with different time constant are also compared. A larger time constant means slower dissolution and release.

- Thus, the pressure of system becomes large if the small time constant is selected as shown. The peak of the system pressure falls within the range between 50 sec to 200 sec but the slope of the change is less steep than all cases.

- Unlike the pressure, there is a large difference in the slope of the temperature change, which seems to be due to neglecting of heat transfer to the wall (heat loss) and interface.



✓ Behavior of pressure at P6



✓ Behavior of temperature at T6

Summary and Conclusion

- The experiment that CO_2 flows into high pressure water was conducted and it was simulated using the in-house code. The in-house code using NC model simulated reasonably well. However, the slope of variation doesn't match exactly due to assumption such as zero heat loss.

- When the above results are considered, it can be concluded that the NC model can be applicable to the phenomenon sufficiently with proper constitutive equations.

Acknowledgement

This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Science, ICT & Future Planning (NRF2016R1A5A1013919).