Preliminary Study on S-CO2 Leakage to High Pressure Water

Jae Jun Lee, Jeong Ik Lee

Department of Nuclear and Quantum engineering, Korea Advanced Institute of Science and Technology (KAIST)
291 Daehak-ro, (373-1, Guseong-dong), Yuseong-gu, Daejeon 34141, Republic of KOREA
*Corresponding author: jeongiklee@kaist.ac.kr

1. Introduction

A Supercritical CO2 (S-CO2) power cycle is considered as the promising power cycle. The main advantages of an S-CO2 power cycle are as follows [1].

1) High efficiency at relatively low temperature
2) Compact components
3) Simple configuration
4) Applicability to various heat sources

Thus, if the intermediate heat exchanger (IHX) fails, CO2 can leak to the primary side. From system behavior to local chemical reactions, many things are dependent on the interaction between S-CO2 and the reactor coolant. Thus, for safety analysis and design of safety features, there are many studies for analyzing the interaction from a safety point of view. However, most studies are for Sodium-cooled Fast Reactor (SFR) and there is limited research on Pressurized Water Reactor (PWR) conditions [2]. In this paper, the experimental and numerical methods for simulating and analyzing the case of PWR are introduced.

2. Experimental Study

In this section, the experimental facility for simulating the phenomenon is introduced briefly because design specification was introduced in the previous work [3]. The experiment conditions and results are also presented.

2.1 Description of experimental facility

As shown in Fig. 3, the facility consists of two tanks and the connection pipe with a nozzle. The content of the experiment is that CO2 flows from a high-pressure tank (CO2, Left tank) to a low-pressure tank (Water, Right tank) through a nozzle. The tap water in the low-pressure tank is pressurized with nitrogen gas. Pressure and temperature of each position are measured for every second. For measurements, eleven resistance temperature detectors and six pressure gauges are installed on the facility as shown in Fig. 4. Leaked CO2 is dissolved into the water and non-dissolved CO2 is accumulated at the top of the tank. The details are shown in Table I.
Fig. 4. Schematic (Top) and Photograph (Bottom) of the experimental facility.

Table I: Design specifications of the experimental facility.

<table>
<thead>
<tr>
<th>Design parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>High/Low pressure tank</td>
<td></td>
</tr>
<tr>
<td>Pressure (MPa)</td>
<td>22</td>
</tr>
<tr>
<td>Temperature (°C)</td>
<td>150</td>
</tr>
<tr>
<td>Volume (L)</td>
<td>47 (I.D : 200 mm, H : 1600 mm)</td>
</tr>
<tr>
<td>Pipe connecting two pressure tanks</td>
<td></td>
</tr>
<tr>
<td>Internal diameter (mm)</td>
<td>57</td>
</tr>
<tr>
<td>Length (mm)</td>
<td>1090</td>
</tr>
<tr>
<td>High pressure tank heater (jacket-type)</td>
<td>Electric capacity (kW)</td>
</tr>
<tr>
<td></td>
<td>5</td>
</tr>
<tr>
<td>Low pressure tank heater (jacket-type)</td>
<td>Electric capacity (kW)</td>
</tr>
<tr>
<td></td>
<td>12</td>
</tr>
<tr>
<td>Valve type</td>
<td>Ball valve</td>
</tr>
</tbody>
</table>

2.2 Experimental conditions

In this paper, two experiment cases are presented. Case 1 is that CO$_2$ flows through the pipe connecting the tanks in the middle. The nozzle diameter is 1.5mm. Case 2 is that CO$_2$ flows through the pipe connecting the tanks from the bottom. The reason for comparing the two cases is to see the effect of CO$_2$ residence time in pressurized water. Table II shows the initial conditions of the tests before opening of the valve.

Table II: Initial conditions of the tests.

<table>
<thead>
<tr>
<th>Case</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>High pressure tank</td>
<td>Pressure (MPa)</td>
<td>10.066 ± 0.003</td>
</tr>
<tr>
<td></td>
<td>Temperature (°C)</td>
<td>76.7 ± 0.3</td>
</tr>
<tr>
<td>Low pressure tank</td>
<td>Pressure (MPa)</td>
<td>1.4413 ± 0.0004</td>
</tr>
<tr>
<td></td>
<td>Temperature (°C)</td>
<td>90.0 ± 0.3</td>
</tr>
<tr>
<td></td>
<td>Mass of N$_2$ (kg)</td>
<td>0.115 ± 0.005</td>
</tr>
<tr>
<td></td>
<td>Mass of water (kg)</td>
<td>38.439 ± 0.408</td>
</tr>
<tr>
<td>Case 2</td>
<td>Pressure (MPa)</td>
<td>9.981 ± 0.003</td>
</tr>
<tr>
<td></td>
<td>Temperature (°C)</td>
<td>75.0 ± 0.3</td>
</tr>
<tr>
<td>Low pressure tank</td>
<td>Pressure (MPa)</td>
<td>1.4163 ± 0.0004</td>
</tr>
<tr>
<td></td>
<td>Temperature (°C)</td>
<td>83.5 ± 0.3</td>
</tr>
<tr>
<td></td>
<td>Mass of N$_2$ (kg)</td>
<td>0.133 ± 0.005</td>
</tr>
<tr>
<td></td>
<td>Mass of water (kg)</td>
<td>37.668 ± 0.411</td>
</tr>
</tbody>
</table>

2.3 Experimental results

In this experiment, dissolved mass of CO$_2$ in the water system is an object to be identified. Dissolved mass can be calculated with assumptions that N$_2$ dissolution is neglected, the dissolved CO$_2$ reaches thermal equilibrium with water and the system pressure is equal to stagnant.

$$
\frac{M_{\text{water}}}{\rho_{\text{water}}} + \frac{M_{\text{N2}}}{\rho_{\text{N2}}} + \frac{M_{\text{dissolved co2}}}{\rho_{\text{apparent}}} + \frac{M_{\text{gas co2}}}{\rho_{\text{co2}}} = V_{\text{tank}} \quad (1)
$$

$$
M_i : \text{mass of component } i
$$
Apparent molar volume of CO\textsubscript{2} is calculated using the model proposed by Hu, Q. et al. (up to 573.15 K, 120MPa) [4]. To evaluate and compare how much CO\textsubscript{2} is dissolved, Duan and Sun solubility predictive model (273-533 K, 0-2000 bar) is used [5]. The mass flow rate of leakage can be calculated using the measured pressure and temperature data of the high pressure tank.

\[
\dot{m} = \frac{V_{\text{tank}}(\rho_{t+\Delta t} - \rho_t)}{\Delta t} \quad (2)
\]

\( \dot{m} \): mass flow rate of CO\textsubscript{2} \\
\( \Delta t \): measurement time interval

Fig. 5 shows dissolved CO\textsubscript{2} mass, non-dissolved CO\textsubscript{2} mass with uncertainty band and fully soluble CO\textsubscript{2} mass according to solubility model for cases 1 and 2.

3.1 Numerical model

When CO\textsubscript{2} is injected into water, the CO\textsubscript{2} jet collapses into fine bubbles. Thus, the authors used the numerical model for calculation of CO\textsubscript{2} dissolution and estimating the bubble size based on mass transfer from a single bubble rising in stagnant water. The up-wind scheme is used and discretized axial length (\(dz\)) is set to 0.005m.

\[
\frac{dM}{dz} = -\frac{KA(C_s - C)}{U} \quad (3)
\]

\( M \): mass of CO\textsubscript{2} bubble \\
\( z \): axial length \\
\( K \): mass transfer coefficient \\
\( A \): bubble surface area \\
\( C_s \): equilibrium concentration \\
\( C \): dissolved concentration \\
\( U \): bubble rising velocity

In this calculation, the bubble rising velocity is assumed to be the terminal velocity of the bubble because the time of accelerating from zero to the terminal velocity is short enough [8]. Terminal velocity and mass transfer coefficient are calculated using models proposed by Tomiyama [9], Higbie [10] and Calderbank [11].

3.2 Numerical results

The below figures show how much bubble dissolves into the water while rising. Fig. 6 shows the calculated results using the model and experimental data for code validation [12]. The height is the axial length that the bubble moves upward. The authors performed preliminary calculations to see some trends under the condition (373.15K, 20MPa). Fig. 7 shows the trends of mass transfer efficiency as a function of bubble size. Fig. 8 shows variation of bubble size as a function of partial pressure related to equilibrium concentration.
4. Summary and Conclusions

The experimental facility and numerical method for simulating and analyzing the case of PWR coupled with the S-CO2 power system are introduced in this paper. The purpose of the experiment is identifying the phenomenon of supercritical CO2 leaking into high pressure water. Using the measured pressure and temperature data, the dissolved mass of CO2 is calculated.

The numerical method is based on the mass transfer from the bubble and it is used to calculate the dissolution according to the bubble size. The bubble size is thought as a key parameter because it is judged appropriate to be used as a model for safety analysis code later.

5. Further works

Eq. 3 does not consider the transient situation. Thus, the model is being developed by considering the variation with time. The model is a combination of the developed models by other researchers. Thus, the results could vary significantly depending on the models. The sensitivity of the model should be studied.

The process of bubble break up is dependent on many conditions such as physical properties, geometry, mass flow rate and so on. However, the authors only considered the physical properties as parameters for estimating the bubble size. It is challenging to consider the geometry effect such as interaction between wall and jet. Flow rate of CO2 can be considered as a factor and can be studied further with more experiments.

ACKNOWLEDGEMENT

This research was supported by the KUSTAR-KAIST Institute, KAIST, Korea.

REFERENCES