The 2025 World Congress on
Advances in Structural Engineering and Mechanics (ASEM25)
BEXCO, Busan, Korea, August 11-14, 2025

CFD-based numerical modeling and validation for preventing condensation in underground utility tunnel

*Sejin Park[1)] Jun-Beom An[2)] Doyeon Lee[3)] and Gye-Chun Cho[4)]

[1), 3) ,4)] *Department of Civil and Environment Engineering, KAIST, Daejeon 34141, Republic of Korea*
[2)] *Korea Institute of Civil Engineering and Building Technology*
[1)] *sejinpark@kaist.ac.kr* [2)] *anssi6@kict.re.kr* [3)] *lionstar21c@kaist.ac.kr*
[4)] *gyechun@kaist.ac.kr*

ABSTRACT

Underground utility tunnels accommodate various types of infrastructure, facilitating efficient use of space in urban areas. Due to their humid environment, it is crucial to establish effective ventilation strategies to ensure safe management and prevent condensation. Condensation poses a significant threat to facilities by accelerating the corrosion of internal equipment, leading to aging and shortened replacement cycles. Moreover, the humid environment caused by condensation negatively affects workers inside the tunnels, as it promotes the growth of bacteria and mold. This study aims to construct a numerical model to suggest strategies to prevent condensation. A CFD-based numerical model was developed using ANSYS Fluent and validated with the collected sensing data and additional field investigation. The results demonstrated that the model accurately reflects real-world conditions. This research serves as a fundamental study for developing a numerical model to predict the condensation phenomenon.

1. INTRODUCTION

Underground utility tunnels, essential components of urban infrastructure, provide a solution to the issue of limited space on the surface. Utility tunnels accommodate multiple facilities, including telecommunications, power, and water supply. Currently, incidents due to the aging of facilities within utility tunnels are continuously increasing, resulting in enormous social and economic costs. Unlike surface-level structure, the interior of the utility tunnels does not have smooth airflow and is subject to high humidity conditions

---

[1)] Graduate Student
[2)] Postdoctoral Researcher
[3)] Ph.D. Candidate
[4)] Professor

such as groundwater infiltration and leakage from deteriorated internal facilities. The concrete structures of utility tunnels are affected by subsurface conditions and tend to maintain lower temperatures than the internal dew point temperature, making them vulnerable to condensation. This condensation promotes corrosion of steel structures, rapid degradation of the materials, and reduced lifespan of equipment such as LED lighting. Furthermore, the growth of mold due to condensation induces a health risk to workers inside the tunnels. Therefore, it is necessary to control the internal environment of the utility tunnels through the operation of ventilation fans. A system capable of assessing the risk for condensation, which is difficult to evaluate using conventional labor-based methods, must be developed. In this study, a CFD-based numerical model for predicting condensation was developed and validated by comparing numerical analysis results and measured data.

2. THEORETICAL BACKGROUND

2.1 Mechanisms of condensation phenomenon

Relative humidity and humidity ratio are key indicators used to represent the moisture content of air. Relative humidity is defined as the ratio of the partial vapor pressure of the air–water vapor mixture to its saturation vapor pressure, as shown in Eq. (1). Saturation vapor pressure is the maximum pressure of water vapor that air can contain at a specific temperature, as described by Eq. (2). Eq. (3) explains the partial vapor pressure, which is the pressure of the water vapor present in the air. It is determined based on the mass ratio (MR) of water vapor to dry air, which reflects the absolute moisture content in the air.

$$\mathrm{RH} = \frac{\mathrm{P}_p}{\mathrm{P}_{\mathrm{sat}}} \times 100\ [\%] \tag{1}$$

$$\mathrm{P}_{\mathrm{sat}} = 610.78 \cdot \exp\left(\frac{17.27 \cdot \mathrm{T_i}}{\mathrm{T_i} + 237.3}\right)\ [Pa] \tag{2}$$

$$\mathrm{P}_p = \frac{MR \cdot 101325}{MR + 0.622}\ [Pa] \tag{3}$$

The humidity ratio refers to the mass of water vapor contained in 1 kg of dry air. Unlike relative humidity, it represents an absolute value that is not affected by temperature, making it suitable for comparing the moisture content in different locations. Condensation generally occurs on concrete walls, which are influenced by the relatively lower underground temperatures compared to the air temperature inside the tunnels. Condensation risk can be assessed by comparing the saturated humidity ratio at the wall temperature, as expressed by Eq. (5), with the humidity ratio of the air near the wall as shown in Eq. (4). Condensation is assumed to occur when the humidity ratio exceeds the saturated humidity ratio for wall temperature (Ma, 2023).

$$\mathrm{HR} = \frac{0.622 \cdot \mathrm{P_p}}{101325 - \mathrm{P_p}} \times 1000\ \ [\mathrm{g/kg_{dry}}] \tag{4}$$

$$\mathrm{HR_{sat}} = \frac{0.622 \cdot \mathrm{P_{sat}}}{101325 - \mathrm{P_{sat}}} \times 1000 \ \ [\mathrm{g/kg_{dry}}] \tag{5}$$

2.2 Governing equations of computational fluid dynamics (CFD)

Computational Fluid Dynamics (CFD) is a numerical method based on governing equations used to understand the influence of fluid flow and heat transfer characteristics within a given domain. In this study, the continuity, momentum, and energy equations were the fundamental governing equations. To represent the motion of the air & water vapor mixture, the convective-diffusion equation was applied. These governing equations were solved according to the finite volume method in the commercial software ANSYS Fluent.

3. NUMERICAL SIMULATION

3.1 Background information of Target utility tunnel

The target utility tunnel section was selected where the condensation frequently occurs during the summer. Total length of the section is 500 meters, and temperature and humidity sensors were installed at five locations throughout the section. Both intake and exhaust fans were installed within the tunnel. The exhaust fan is located approximately 400 meters away from the intake fan. However, since the exhaust fan was not in operation during the data collection period, its influence was excluded during the model validation process.

3.2 Meshing

Meshing is the process of dividing the whole domain into discrete volumes to solve the governing equations for fluid flow analysis. In this study, a tetrahedron meshing method was utilized with an element size of 0.13 m, resulting in a total of 8,366,516 elements. To assess the quality of the mesh, the minimum orthogonal quality and maximum skewness were evaluated, with values of 0.2 and 0.8, respectively, indicating a ‘good’ quality mesh. In general, the minimum orthogonal quality should be greater than 0.1, and the maximum skewness should be less than 0.95.

3.3 Boundary conditions and solver settings

To validate the numerical model compared to collected temperature and humidity data, additional field investigations were performed to determine the specific conditions of the flow characteristics of the target utility tunnel. Additionally, Observations of actual condensation areas were also measured to validate the numerical model. The air inlet boundary condition was set as a velocity inlet, and air outlets were defined as a pressure outlet. Temperature and humidity values were assigned based on sensor data before the activation of the intake fan. Since wall temperature is a critical factor in determining the occurrence of condensation, wall temperatures were measured at 10-meter intervals and reflected in the model. For the viscous model, the standard k–ε model, which is appropriate for general fluid flow, was applied (Seong 2017). The species transport model was adopted to simulate the mixture of water vapor and air within the computational domain. SIMPLE algorithm was used for pressure–velocity coupling, second-order upwind and second-order schemes were applied for momentum and pressure discretization, respectively. During the simulation, convergence was considered to be achieved when the residuals for continuity, velocity, k, ε, and H_2O fell below 1×10^{-3}.

4. Results

As a result, the numerically derived temperature and humidity values at four locations were within 10% error compared to the actual sensor data measured three hours after the intake fan was activated. Additionally, through post-processing, the difference in humidity ratio (ΔHR) between the saturated humidity ratio corresponding to the wall temperature and the humidity ratio near the wall surface was calculated for each area. Condensation is considered to occur in regions where ΔHR is negative. The distribution of condensation areas derived from the numerical model showed a similar trend to the actual condensation occurrence observed in the field investigation. Therefore, the developed condensation prediction model was determined to be reliable.

Measured points	A		B		C		D	
Results	Numerical results	Sensor data	Numerical results	Sensor data	Numerical results	Sensor data	Numerical results	Sensor data
Temperature [℃]	25.43	27.01	25.74	24.7	25.61	24.11	25.56	24.07
Humidity ratio $[\mathrm{g/kg_{dry}}]$	19.99	20.65	17.99	20.19	18.11	20.12	17.77	16.85

Table 1. Comparison of numerical analysis results and measured sensor data

5. CONCLUSION

A CFD-based numerical approach was conducted to predict condensation occurrence inside a target underground utility tunnel. In this study, the numerical model was successfully validated using sensor measurements and additional field investigations. The model accurately predicted the temperature and absolute humidity after the operation of the intake fan and successfully identified the areas where condensation occurred. Based on the validated model, ventilation strategies will be developed considering both internal and external thermal and humidity conditions of the utility tunnel. This research will provide a fundamental basis for the prediction and management of the condensation phenomenon in advance.

REFERENCES

Ma, H., Zhou, X., & Huang, J. (2023), “Effect of ventilation on thermal and humidity environment of the underground utility tunnel in the plum rain season in southern China: Field measurement and CFD simulation”, Underground Space, 13, 301-315.

Seong, N. C., Kim, J. H., & Choi, K. B. (2017), “CFD analysis of temperature and relative humidity distribution as air flow rate variation in the underground utility pipe tunnel”, Journal of Korean Institute of Architectural Sustainable Environment and Building Systems, 11(4), 273-282.