3D RADON DISTRIBUTION MODELLING IN THE TYPICAL KOREAN HOUSE BASED ON CFD CODE

Sy Minh Tuan Hoang[1], Gwang Min Sun[1*], Jiseok Kim[1], Han Rim Lee[1], Hani Baek[1]

[1]*Korea Atomic Energy Research Institute, 111 Daedeok-Daero 989 Beon-Gil, Yuseong-gu, Daejeon, Republic of KOREA*
**Email Address: gmsun@kaeri.re.kr*

The concentration of radon inside the complex building was simulated based on the Computational Fluid Dynamics (CFD) programs as ANSYS FLUENT and OpenFOAM. The model of the building in this study is a typical Korean house with two-floor structure. Using the CFD programs to solve the governing equations incorporation with the radon sources, the impact of radon ventilation on indoor air quality was studied in this paper.

I. INTRODUCTION AND METHOD

I.A. Introduction

Radon is one of the major and most harmful indoor air pollutants that occurred in the environment and many countries threatens peoples' health, whereas radon risk is known as the second causes of lung cancer after cigarette smoking (Ref. 1). In standard conditions, characteristics of radon are as shown in Table I. Some amount of radon and radon daughters are present everywhere in the soil, water, and air. Particularly, high radon levels occur in regions where the soil or rock is rich uranium. Radon in the soil can be drawn into the building and can accumulate to high levels. For reducing the radon concentration in a residential building at the standard level, forced ventilation is usually used.

TABLE I. Physico-chemical characteristics of radon (Keith S, 2012)

Density (kgm^{-3})	Diffusivity in Air (m^2s^{-1})	Decay rate (s^{-1})	Atomic weight ($gmol^{-1}$)
9.73	1.1×10^{-5}	2.1×10^{-6}	222

In this study, the impact of ventilation on indoor radon was studied by using Computational Fluid Dynamics (CFD) to achieve indoor air quality and energy efficiency. Currently, measurement techniques and standards and regulations of indoor pollutants and ventilation, particularly related to indoor radon cannot be able to provide a secure, safe and energy efficient indoor climate. This is why the indoor airflow distribution is very complex, and with changing building geometry and operation condition, the treatment of airflow pattern substantially will be changed, whereas the rules are usually independent of the building's features. Furthermore, the indoor standards and regulations are based on the average amount of pollutants in a room, whereas the pollutant distributions are not identical and are varied throughout the room. Then the current techniques are not exactly valuable and acceptable. This study presents results from simulation studies on ventilation and radon mitigation in residential buildings, in viewpoints of indoor air quality. The CFD technique is applied to predict, visualize and calculate of mixture radon-air flow.

I.B. Method

In this study, CFD analysis is used to investigate the effects of indoor air parameters for indoor radon treatment numerically. The CFD can be easily used for indoor air flow analysis and reduces the mathematical limitations. The CFD programs were used to simulate and predict of radon distribution in the residential buildings as ANSYS FLUENT (Ref. 2) and OpenFOAM (Ref. 3). The case house in this study is a typical detached house in KOREA. This house is a two-floor structure, and the area of each floor is about 100m^2 with the basement is located in the bedrock. The sources of radon enter into the house are presented in Figure I.

CFD programs are designed to solve the governing equations in the form of partial differential equations and are employed to solve equations involving velocity, temperature and species transport numerically. One of the simulated results is presented in Figure II.

The mass conservation Eq. (1):

$$\frac{d\rho}{dt} = \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial t}(\rho u) + \frac{\partial}{\partial t}(\rho v) + \frac{\partial}{\partial t}(\rho w) = 0 \qquad (1)$$

where ρ is density and u, v and w are the velocity components of the velocity vector V in the x, y, and z directions respectively (Ref. 4).

The momentum conservation Eq. (2):

$$\frac{\rho dV}{dt} = -\nabla P + \mu \nabla^2 V + F_B \qquad (2)$$

where P is static pressure vector, μ is air viscosity and F_B is body force vector.

The energy conservation Eq. (3):

$$\frac{\partial(\rho c_p T)}{\partial t} + \nabla(\rho c_p T V) = \nabla(K \nabla T) + S_e \qquad (3)$$

where C_p is the specific heat capacity of the air, k is the thermal conductivity, and Se is an energy source, which is zero in this case (Ref. 4).

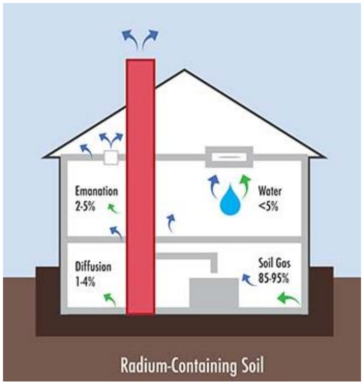

Fig. I. Sources of radon enter to the house.

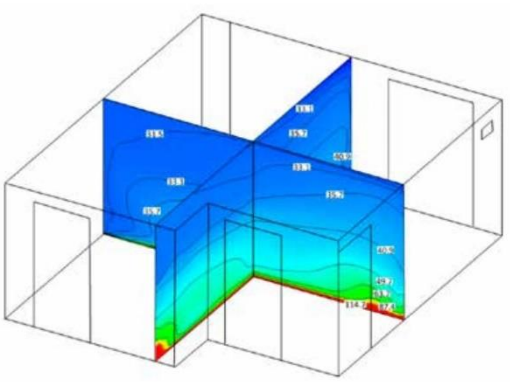

Fig. II. The concentration of radon (Bqm^{-3}) in two vertical planes in the first floor of the house.

II. CONCLUSIONS

The modeling method of radon in CFD programs is the key factor for solving transport equations in turbulence modeling and enhancing the accuracy of the results. The CFD programs (ANSYS FLUENT and OpenFOAM) are a useful and applicable tool for investigating these factors individually, and in this study, it is used as a standalone and available tool to simulate and predict the radon level and distribution in complex building buildings.

REFERENCES

1. https://www.epa.gov/radon, (Accessed January 2017).
2. http://www.ansys.com/Products/Fluids/ANSYS-Fluent, (Accessed January 2017).
3. http://www.openfoam.com/, (Accessed January 2017).
4. F. M. White, “Fluid Mechanics”, McGraw-Hill, New York, USA (2011)