TEMPERTURE AND FLOW ANALYSIS IN AN OVEN USING COMPUTATIONAL FLUID DYNAMCIS

Wonwoo Jeon[1], Phil Kim[1], Eunseop Yeom*[1,2]

1. Graduate School of Mechanical Engineering, Pusan National University

2. School of Mechanical Engineering, Pusan National University

Busan, 609-735, Republic of Korea

** E-mail: dygks5741@pusan.ac.kr*

ABSTRACT

Ovens can use a pyrolytic self-cleaning mode that operates at high temperatures. This can cause external heat concentration and affect internal uniformity due to strong natural convection. This study uses transient Computational Fluid Dynamics (CFD) analysis with the DO radiation model and full buoyancy settings to better simulate heat transfer and airflow, utilizing the Realizable k-ε turbulence model. Results show a maximum error of 80°C and minimum error of 11°C providing insights for reducing heat concentration and verifying through simulation that the temperature increases to the level required for pyrolysis.

KEY WORDS

Oven, Temperature distribution, Computational fluid dynamics

1. INTRODUCTION

The self-clean mode of ovens utilizes pyrolysis to remove food residue from the oven's interior. Figure 1 shows the interior of an oven during cleaning in pyrolysis mode. Pyrolysis refers to a chemical process in which organic matter decomposes at high temperatures without oxygen or other oxidizing agents. Since it operates without oxygen, pyrolysis has the advantage of not producing oxides or dioxins, which are harmful to the environment and human health. However, because it operates at high temperatures, this process consumes significant energy and can cause heat concentration on the oven's exterior, posing a risk of injury to users. It is thus essential to evaluate the temperature and flow fields within the oven under pyrolysis mode and to address potential issues.

Computational fluid dynamics (CFD) has been applied to visualize and optimize oven temperature and flow fields. Mistry, H.[1] compared the accuracy of different turbulence and radiative heat transfer models to simulate oven interior temperature and flow fields accurately. Smolka, J.[2] evaluated the uniformity of heat distribution within the oven based on heater location and configuration, as well as fan placement, using CFD. Park, S.[3] increased heat distribution uniformity inside the oven by designing an optimal fan casing shape using CFD. However, these studies are limited by their use of steady-state analyses, which do not account for the hysteresis control where the heater cycles on and off sequentially.

In this study, we conduct a transient CFD analysis of the oven interior to observe changes in temperature and flow fields over time and verify that the oven's interior temperature increases to a level sufficient for pyrolysis reactions to occur. The resulting temperature distribution on the oven door glass provides foundational data for addressing heat concentration issues in the oven door system.

2. EXPERIMENTAL AND NUMERICAL SET UP

2.1 Oven temperature measurement experiment

To validate the accuracy of our analysis, we measured the internal temperature of a built-in electric oven (LG Electronics, LSEL6335f) equipped with electric heaters to compare between experiment and simulation results. Figure 1(a) shows the overall structure of the oven. Using a K-type thermocouple, we recorded the temperature at the center of the oven interior at 5-second intervals. The oven operates with a hysteresis control system, alternating between the Bake heater located at the top and the Broil heater at the rear center. For uniform temperature distribution, the fan operates at 1600 RPM, and the fan case is located around the fan and the bake heater. The oven cavity is surrounded by insulation to prevent heat transfer with the external environment.

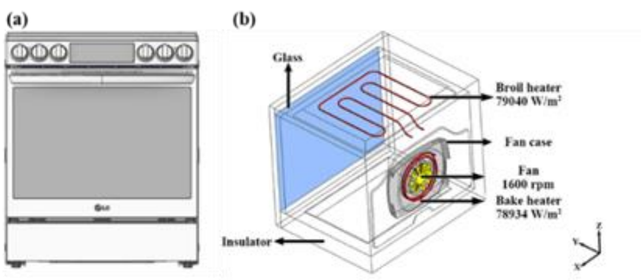

Fig.1: (a) Electric built-in oven (b) Schematics of simulation

2.2 Computational Fluid Dynamics (CFD) Analysis of the Oven

This study utilizes Ansys Fluent 2022R1 for the analysis. Figure 1(b) shows the oven geometry and conditions used in the simulation, which includes the Bake heater, Broil heater, and a fan rotating at 1600 RPM. The fan and Bake heater are encased within a fan casing, while the entire oven is insulated. To accurately capture the complex internal flow structure, the Realizable k-ε turbulence model was applied. Due to significant temperature variations in the internal air, the air properties, such as density and viscosity, were input as a temperature-dependent function using a piecewise-linear approach. Additionally, to account for the changing density of the air in buoyancy calculations, the buoyancy effect-full option was used. The Discrete Ordinates (DO) radiation model was used to accurately model semi-transparent properties of the oven door glass. The heaters are controlled using hysteresis control. To simulate the interval operation of the heaters, a user-defined function was implemented, allowing each heater to alternate between operation and stoppage based on a specific interval time.

3. CONCLUSION

The transient analysis was conducted with a time step of 1 second, for a total duration of 3030 seconds. In the CFD analysis results, the oven center point temperature is the average temperature of the cells within a cube with 10 mm edges, located at the center, and was collected using a user-defined function. Figure 2 shows the validation results based on the temperature at the central part of the oven interior. Figure 2 shows a maximum error of 80°C and a minimum error of 11°C. Additionally, Figure 2 shows fluctuation, indicating that the alternating operation of the heaters was accurately simulated using the user-defined function (UDF).

Figure 3 illustrates the flow lines inside the oven at 3030 seconds. Relative to the fan, flow lines predominantly appear from the top on the left side and from the bottom on the right side, which is due to the fan casing. Holes in the top left and bottom right of the fan casing allow airflow from the fan

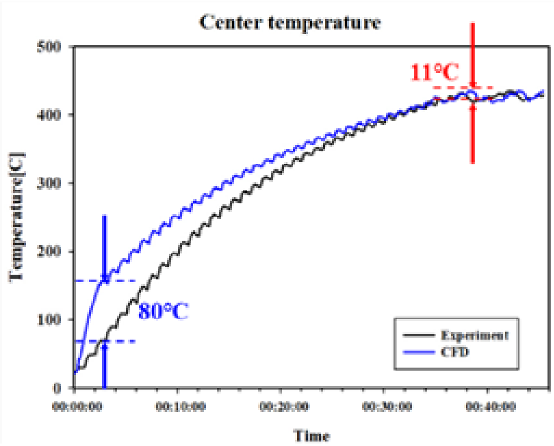

Fig.2: Validation results using center temperature

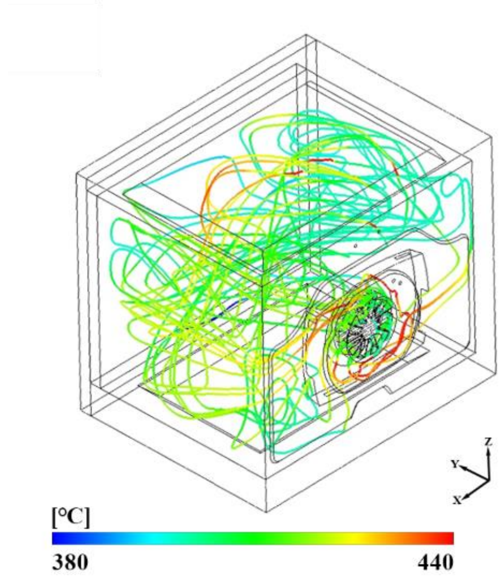

Fig.3: Streamline within the oven at 3030sec

to exit into the oven interior. Due to the fan's rotation, the flow exits more forcefully from the right hole, causing the circulating flow within the

Oven to persist longer. As the air circulates near the heater, it absorbs heat, resulting in a higher temperature in the flow exiting from the right hole.

Finally, the inner wall temperatures of the oven all reached above 380°C, confirming that pyrolysis reactions can occur within the oven.

ACKNOWLEDGEMENTS

This research was supported by the National Research Foundation of Korea, funded by the government (Ministry of Science and ICT) (No. 2020R1A5A8018822).

REFERENCES

1 Mistry, H., Dey, S., Bishnoi, P. and Castillo, J. L., 2006, "Modeling of Transient Natural Convection Heat Transfer in Electric Ovens," *Applied thermal engineering*, Vol. 26, No. 17-18, pp. 2448-2456.

2 Smolka, J., Nowak, A. J. and Rybarz, D., 2010, "Improved 3-D Temperature Uniformity in a Laboratory Drying Oven Based on Experimentally Validated Cfd Computations," Journal of Food Engineering, Vol. 97, No. 3, pp. 373-383.

3 Park, S. H., Kim, Y. H., Kim, Y. S., Park, Y. G. and Ha, M. Y., 2018, "Numerical Study on the Effect of Different Hole Locations in the Fan Case on the Thermal Performance inside a Gas Oven Range," Applied Thermal Engineering, Vol. 137, No., pp. 123-133.