Model of Thermal Plume above Cooking Gas Stove for Designing Ventilation

Yuki Shimanuki¹, Takashi Kurabuchi², Yoshihiro Toriumi³, and Yasuhisa Asawa¹

¹Tokyo Gas Co., Ltd, Tokyo, Japan

²Department of Architecture, Faculty of Engineering, Tokyo University of Science, Tokyo, Japan

³Division of Architecture, Civil and Environmental Engineering School of Science and Engineering, Tokyo Denki University, Saitama, Japan

Abstract. A model of the thermal plume above a cooking gas stove using computational fluid dynamics (CFD) analysis was studied to predict the heat and vapor released during cooking. The combustion gas released from the burner installed in the gas stove was considered as air in which thermal energy was adjusted so that the thermal plume above the gas stove could be simulated. Therefore, the model could predict the thermal plume above the gas stove based on the capacity of the burner and pot size. For validating the simulated flow fields, the results of the velocity distributions above the gas stove calculated using CFD analysis models were compared with the results of the velocity distributions measured with particle image velocimetry (PIV). In conclusion, the analysis results were in good agreement with the measurement results. However, the velocity in the vertical direction calculated using CFD above the center of the burner was higher than the velocity measured using PIV along the axis from the center of the burner.

1 Introduction

Computational fluid dynamics (CFD) analysis is often used to design ventilation in commercial kitchens. The heat and vapor released during cooking negatively affect the environment in the kitchen. Therefore, appropriate ventilation must be designed to balance environmental degradation with efficient utilization of energy. For designing appropriate ventilation, CFD analysis can be used to calculate the capture efficiency of the hood for different types of kitchen appliances and different disturbance conditions.

Several CFD models for calculating the capture efficiency of the exhaust gas released from the gas stove have been studied [1-3]. In these models, the measured data of the velocity distribution corresponding to the size of each burner and pot is needed to set the boundary conditions. Omori et al. proposed the exhaust gas plume model [4]. The combustion gas released from the burner installed in the gas stove was considered as air in which thermal energy was adjusted so that the thermal plume above the gas stove could be simulated. Therefore, the model could predict the thermal plume above the gas stove without the measured data of velocity distributions. However, it is not verified whether the velocity distribution corresponding to the pot shape can be simulated.

Hence, in this study, a model of the thermal plume above a cooking gas stove based on the Omori's model [4] was developed for different pot diameters. For validating the simulated flow fields, velocity distributions

above a commercial cooking gas stove were measured using particle image velocimetry (PIV). A gas stove equipped with a 14.5-kW burner was placed in a test room composed of a ventilation system that guaranteed undisturbed flow from the system. The cooking mode was evaluated with boiling water in the pot. The results of velocity distributions above the gas stove calculated using CFD analysis models were compared with the results of velocity distributions measured using PIV. In conclusion, the analysis results showed good agreement with the measurement results.

2 Nomenclature

 Q_{in} : Heating power of the burner Q_{trn} : Heat transfer through the pot Q_{tab} : Heat transfer through the gas stove Q_{ex} : Heat transfer across the top of the pot Q_{eff} : Heat transfer through the pot Q_{pan} : Radiant heat transfer energy of the pot η : Thermal efficiency of the gas stove A_{pan} : Surface area of the pot ε_I : Emissivity of the pot ε_2 : Emissivity of the wall σ : Stefan–Boltzmann constant (5.67 × 10⁻⁸ Wm⁻²K⁻⁴) T_p : Surface temperature of the pot h_{tab} : Heat transfer coefficient A_{tab} : Surface area of the top of the gas stove t_{tab} : Surface temperature of the top of the gas stove t_0 : Room temperature

Yuki Shimanuki: n-yuki@tokyo-gas.co.jp

https://doi.org/10.1051/e3sconf/201911101030

dt: difference between combustion gas temperature and room temperature *d*: Diameter of the pot

3 Analysis methods used in CFD

3.1 Configuration of calculated field

Figure 1 shows the calculation domain for the CFD simulation. The pot and gas stove were set in the center of the calculation domain. We used half of the area as the calculation domain because the two halves of the calculation area were symmetrical. Air was supplied from the floor and was exhausted from the ceiling. The dimensions of the pot and the gas stove are shown in Figure 2. We used three pot diameters of 330 mm, 390 mm, and 440 mm to evaluate whether the model could estimate the effect of pot size on the thermal plume above the gas stove.





Fig. 1. Calculation Domain for CFD Analysis.

Fig. 2. Dimensions of Pot and Gas Stove.

3.2 Model of combustion gas

We considered the gas released from the gas stove as air in which thermal energy was adjusted to simulate the thermal plume above the gas stove.

Figure 3 shows the model used for representing the combustion gas. The natural gas passing through the burner burns transmits heat to the pan and gas stove and flows over the pan.



Fig. 3. Schematic of the Model.

The thermal energy of the air supplied from burner was adjusted to match the heat flux across the top of the pot. The heat flux across the top of the pot can be described as

$$Q_{ex} = Q_{in} - Q_{trn} - Q_{tab.} \tag{1}$$

Heat transfer across the top of the pot is described as heat transfer through the pot and radiant heat transfer energy as

$$Q_{ex} = Q_{in} - Q_{eff} - Q_{pan} - Q_{tab}.$$
 (2)

Heat transfer through the pot, radiant heat transfer energy of the pot, and heat transfer through the gas stove can be described as follows:

$$Q_{eff} = \eta \, Q_{in} \tag{3}$$

$$Q_{pan} = A_{pan} \ \varepsilon_{1} \varepsilon_{2} \ \sigma_{4} T_{p} \tag{4}$$

 $Q_{tab=} h_{tab} A_{tab}(t_{tab} t_0) \tag{5}$

Table 1 presents the analysis cases. Each parameter was determined in correspondence with the gas stove used for measurements in the PIV tests and heating value of natural gas in Japan. The heating thermal efficiency of the gas stove was measured using the pot with diameters of 330 mm, 390 mm, and 440 mm in advance.

Table 1. Analysis cases

| | d | mm | 330 | 390 | 440 |
|----|------------------|----------------------------------|-------|-------|-------|
| | Qex | W | 7840 | 6570 | 5710 |
| | Q_{in} | W | 13900 | 13000 | 13000 |
| 11 | $Q_{e\!f\!f}$ | W | 4460 | 5690 | 6190 |
| | Q_{pan} | W | 193 | 247 | 297 |
| 11 | Q_{tab} | W | 460 | 460 | 460 |
| | η | % | 31.4 | 39.5 | 43.0 |
| | A_{pan} | m ² | 0.282 | 0.352 | 0.414 |
| | A_{tab} | m ² | 1.00 | 1.00 | 1.00 |
| | E1, E2 | - | 0.900 | 0.900 | 0.900 |
| | h_{tab} | Wm ⁻² K ⁻¹ | 10 | 10 | 10 |
| | T_p | °C | 100 | 100 | 100 |
| | t _{tab} | °C | 70 | 70 | 70 |
| | to | °C | 24 | 24 | 24 |

3.3 Boundary conditions

Table 2 lists the boundary conditions. We set a boundary condition of floor as velocity inlet to stabilize the calculation. The boundary condition for the burner was set based on the model of the combustion gas.

Table 2. Boundary conditions

| D 1 | T | | |
|------------|--|--|--|
| Boundary | Туре | | |
| | Velocity inlet | | |
| Floor | (Temperature: 24.0 °C, Velocity: 0.0400 m/s, | | |
| | Density: 0.950 kg/m ³ , Material: Air) | | |
| Cuiling. | Pressure outlet | | |
| Celling | (Pressure: 0.00 Pa) | | |
| W-11 | Slin 11 A dishatia | | |
| Wall | Slip wall, Adiabatic | | |
| | Velocity inlet | | |
| Burner | (Temperature: 100 °C, Velocity: 2.87 m/s, Density: | | |
| | 0.950 kg/m ³ , Material: Air) | | |
| | Velocity inlet | | |
| Surface of | (Temperature: 100.0 °C, Water vapor generation | | |
| Water | rate: 0.0330 m/s, Density: 0.598 kg/m ³ , Material: | | |
| | H ₂ O) | | |
| Surface of | Wall | | |
| the pot | (Temperature: 100.0 °C) | | |

3.4 Code and mesh

Table 3 gives the analysis conditions. The mesh size was 10 mm, and the number of cells was 560,000.

| Т٩ | ahle | 3 | Δna | lysis | cond | itions |
|----|------|----|------|-------|------|--------|
| 14 | aDIC | э. | Alla | 19515 | conu | nuons |

| Software | Star-CCM 12.01 | |
|------------|--------------------|--|
| Turbulence | Standard k–ε Model | |
| Density | Ideal gas | |
| Algorism | Steady State | |

4 Measurement methods for velocity distributions and thermal distributions

4.1 Test room for measurement

A gas stove (900 mm \times 600 mm with a height of 800 mm; RSB-096SV/RINNAI) was set at the center of the test room. The convection plumes from a 14.5-kW burner were studied. The cooking mode of the stove was set with boiling water in pots with different diameters of 330 mm, 390 mm, and 440 mm. For velocity distributions measurement, two pot diameters of 330 mm and 390 mm were used. For thermal distributions measurement, three pot diameters 330 mm, 390 mm, and 440 mm were used.

4.2 Measurement methods for PIV test

Figure 4 shows the schematic of the methods used to measure the velocity distributions. Two different regions were set for PIV measurements to reveal the detailed development of the thermal plume. A vertical crosssection, two-dimensional PIV test was performed. The shooting area was 600 mm × 800 mm. For a horizontal cross-section at a height of 1000 mm from the top of the gas stove, a stereo PIV test was performed. The plume in the region directly affects the capture efficiency because ventilation hoods for gas stoves are set at this height in commercial kitchens. Therefore, we measured the detailed velocity composition in this region. The shooting area was 600 mm \times 800 mm. The velocity plumes were fully developed at a height of 1000 mm from the top of the gas stove. The frame straddling technique was used for measurement. A double-pulse laser (LS-2145PIV;



LOTIS TII, Minsk, Belarus) and double-shutter camera (FtrNPC; Flowtech Research Inc., Yokohama, Japan) equipped with a 50-mm lens (AF Micro-Nikkor; Nikon, Tokyo, Japan) were set up. The laser and camera were synchronized with a pulse generator (VSD2000; Flowtech Research Inc.) and software (FtrPIV Ver.3.2; Flowtech Research Inc.) to record matched pairs of particle images. The laser frame straddle time was adjusted to ensure that the tracer particle moved no more than 5 px during this time; consequently, it was set to 1300 µs for the vertical cross section and 500 µs for the horizontal cross section. The laser-pulse frequency was 10 Hz, which was the maximum value permitted by the device. The statistical data comprised 2000 images from the two-dimensional PIV test and 4000 images from the stereo PIV test. The result was arranged into a time-averaged flow field. The airflow was supplied from the displacement ventilation (FLOORMASTER; system Takasago Thermal Engineering Co., Ltd., Tokyo, Japan), which guaranteed an undisturbed convection flow from the ventilation system. Olive oil mist was selected as the tracer particle. Olive oil mist particles were released from three oil mist generators in the cooking mode.

4.3 Measurement methods for thermal distributions

The difference between the plume temperature at a height of 1000 mm from the top of the gas stove and room temperature was measured with hot wire anemometers. The measurement was not performed simultaneously with PIV because the anemometers would disturb the fluid velocity. The flow rate of the input gas was measured with a mass flow meter (CMG500; Azbil, Fujisawa).

5 Results and discussion

5.1. Comparison of the CFD results and experimental results



(a) Two-dimensional PIV test for vertical cross section (b) Stereo PIV test for horizontal cross section **Fig. 4.** Schematic of methods used to measure velocity distributions with PIV (u is the velocity component in x direction, v is the velocity component in y direction, and w is the velocity component in z direction).



Fig. 5. Vertical velocity distributions at heights up to 1000 mm from the top of the gas stove with the 390-mm-diameter pot (the filled triangles indicate experimental results from the left half of the image; the unfilled triangles indicate experimental data from the right half of the image, and the solid line represents calculated results).

Figure 5 shows a comparison of calculated results and experimental results of the vertical velocity distributions at heights up to 1000 mm from the top of the gas stove. The calculated results were in good agreement with the measurement results. However, the velocity in the vertical direction calculated by CFD above the center of the burner was higher than the velocity measured by PIV along the axis from the center of the burner. Kiyosuke et al. suggested that standard k- ϵ models predict the insufficient diffusion in the buoyant plume, so the simulated capture efficiency of exhaust gas released from the gas stove tends to be overestimated [5]. Therefore,

investigating the turbulence model for predicting the thermal plume is necessary in future works, to predict the capture efficiency accurately.

5.2 The velocity and temperature distributions above the pot

Figure 6 shows a comparison of the calculated results and experimental results for the temperature rise and velocity distributions with different diameters of the pot. The results were compared at a height of 1000 mm from the top of the gas stove. The ventilation hood is generally installed at this level. Thus, the thermal plume at this



Fig. 6. Comparison of calculated results and experimental results of the velocity distributions and temperature rise with different diameters of the pot.

height affects the capture efficiency. The profile of the temperature rise calculated using CFD agreed with the profile obtained experimentally regarding the effect of the diameter of the pot on the thermal plume. However, the velocity in the vertical direction calculated using CFD above the center of the burner was also higher than the velocity measured using PIV along the axis from the center of the burner.

6 Conclusions

A model of the thermal plume above the cooking gas stove using CFD analysis was studied based on the present study to predict the heat and vapor released during cooking.

The analysis results calculated were in good agreement with the measurement results obtained via the PIV test. However, the velocity along the vertical direction calculated using CFD above the center of the burner was higher than the velocity measured using PIV along the axis from the center of the burner. Therefore, investigating the turbulence model for predicting the thermal plume is necessary in future works to predict the capture efficiency accurately.

Acknowledgements

This study was made possible by the technical support from Dr. Toshiaki Omori of the University of Tokyo. We would also like to thank Reo Murakami and Masaru Shimokawa of the Fundamental Technology Research Institute of Tokyo Gas Co., Ltd for their help with the simulation. Moreover, we would like to thank Flowtech Research Inc., Yokohama, Japan, who conducted the examination of PIV test.

References

- Y. Kondo, Y. Abe, A. Miyafuji, Y. Aizawa, K. Akagi, J. Environ. Eng. **73**1383 (2008).
- 2. T. Momose, T. Yamanaka, H. Kotani, A. Yamamoto, J. Environ. Eng. 68, 49 (2003).
- 3. J. Sakaguchi et al. Summaries of technical papers of Annual Meeting of AIJ. **D-2**, 673 (2008).
- 4. T. Omori, Y. Shimanuki, M. Aibara, A. Okuda, Trans. SHASE, 13 (2013).
- 5. H. Kiyosuke, T. Kurabuchi, Y. Toriumi, Y. Shimanuki, Y. Asawa, Trans. SHASE, 49 (2018).