STUDY ON REPRODUCIBILITY OF EXHAUST DIFFUSION PROPERTIES OF GAS WATER HEATERS

Saki Nakano [a], *Takashi Kurabuchi* [b], *Jeongil Kim* [c].

[a] Department of Architecture, Graduate School of Engineering, Tokyo University of science, Tokyo, Japan, mbcaj.10.fk27do30@gmail.com.

[b] Department of Architecture, Graduate School of Engineering, Tokyo University of science, Tokyo, Japan, kura@rs.tus.ac.jp.

[c] Department of Architecture, Graduate School of Engineering, Tokyo University of science, Tokyo, Japan, jeongil.kim@rs.tus.ac.jp.

Abstract. When installing a gas water heater in a common corridor, the common way of the installation in Japan, there are restrictions on the installation location and the shape of the corridor in order to avoid the exhaust gas stagnating in the corridor and flowing into the rooms through the vent. Therefore, the standards limit the building planning and require more relaxation. We had been conducting experiments by using full-scale models over the past years, however, as there are many shapes of common corridor, it is difficult to examine all the possibilities. It is necessary to reproduce the exhaust properties on computational fluid dynamics (CFD) so that the examination of reference relaxation can be performed by simulation using CFD. We needed to collect target data to confirm reproducibility by actual measurement. It was ejected without any obstacles to the jet flow, and we measured the velocity, temperature, CO2 concentration, etc.
We estimated that the model of the interface on a gas water heater and setting boundary conditions can be the factors to improve the reproducibility.

Keywords. heater, corridor, exhaustion, prediction
DOI: https://doi.org/10.34641/clima.2022.336

1. Introduction

In Japan, we have a bath culture and home water heater are popular. There are two types of air supply and exhaust, we usually use Roof Top Flue gas water heater in a common corridor of apartments.

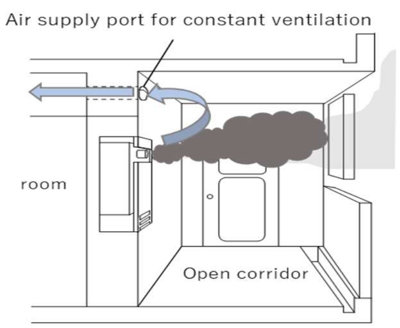


Figure1. Installation status of gas water heaters in Japan

In such cases, installation standards have been established to prevent exhaust gases from stagnating in the corridor and flowing into the rooms through the vents. Installation standards restrict the location of water heaters and vents, and the shape of corridors.

Since the current standards are a constraint on architectural planning, we have been conducting various experiments using full-scale models in order to relax the standards as much as possible. However, in recent years, heaters and architectural styles have diversified, and it is difficult to study all cases by experiment.

Therefore, we are trying to establish a method to reproduce the exhaust gas in CFD to enable us to decide whether or not to install the water heater by simulation analysis. If we can examine the possibility of installation using CFD, it will be easier to determine the installation of various shapes, which will expand the range of design.

2. Measurement of exhaust jet

An image of a gas water heater and an air supply port installed in an open corridor is shown below.

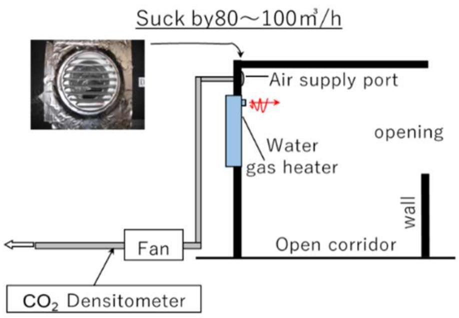


Figure2. a gas water heater and an air supply port installed in an open corridor

We have attempted to reproduce the exhaust in previous studies, but adequate methods have not yet been established. We believe the reason for that is that they were using the CO2 concentration in the vents to check for consistency. In this study, we operated the water heater in a space where there was nothing that could affect the exhaust air, and measured the temperature, CO2 concentration, and other exhaust air properties. The results will be used to check the consistency with the analysis results to improve reproducibility.

2.1 Measurement Summary

We measured the velocity, temperature, and concentration at the exhaust port as shown in the figure3.

For the measurement of exhaust distribution, a mesh with 56 thermocouples uniformly arranged so as not to interfere with the exhaust air is used. By changing the mesh height in two steps, we measure the temperature at all 98 points within 0.8 m above and 0.4 m below the exhaust port height. A tube is placed at 18 of these points to measure the CO_2 concentration. By moving the above mesh parallel to the gas appliances, the diffusion of exhaust gas at points 1m, 2m, and so on away from the appliances was measured.

Figure3. Experimental method for blowout condition of Exhaust port

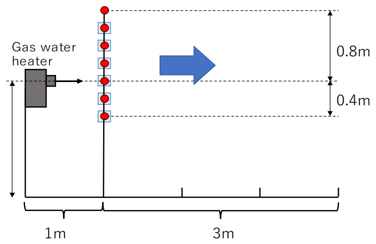


Figure4. Experimental method for exhaust distribution

In addition, we conducted visualization experiments to confirm the trajectory of the vertical cross section in the blowing direction.

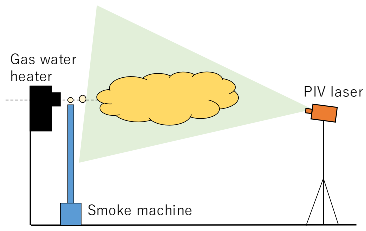


Figure5. Experimental method for visualization of exhaust

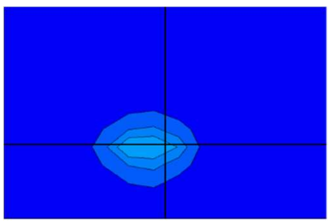
2.2 Measurement results

We focused on two different operating conditions: "maximum combustion" and "heating only. The case of maximum output in heating and hot water supply is called "maximum combustion", and the state in which only heating is operated is called "heating only". The exhaust measurement results for each condition are shown below.

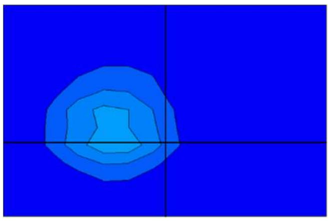
Table1. The exhaust measurement results

	Maximum combustion	**Heating only**
Velocity[m/s]	4.81	3.08
Temperature[K]	341.3	324.8
Concentration[ppm]	65005	12751

Depending on the output, we found differences in temperature, CO2 concentration and ascent rate, as well as left and right deflections.

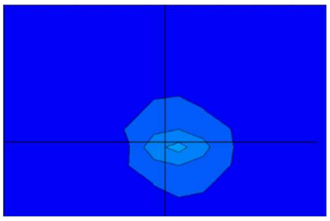


at *1.0m* from the equipment

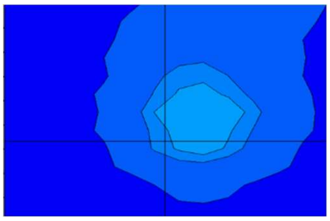


at *2.0m* from the equipment

Figure6. “*Maximum combustion*” the temperature distribution



at *1.0m* from the equipment



at *2.0m* from the equipment

Figure7. “heating only” the temperature distribution

The internal shape of the exhaust outlet was checked, and it was found that the outlet surface was divided between heating and hot water supply, causing deflections to be created.

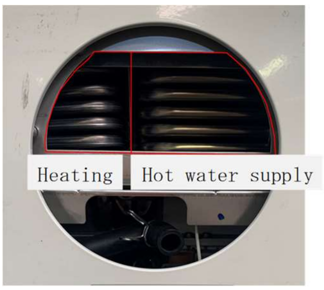


Figure8.interal shape of the exhaust outlet

3. CFD summary

We set up an analysis area of 16 m wide, 10 m deep, and 10 m high was set up. And we installed an exhaust outlet at a height of 2.2 m to reproduce the experimental conditions. As mentioned earlier, it was found that the exhaust air blowing surface is divided between the heating system and the hot water supply system, so an adapter shape that reproduces the actual situation is used. At First, we tried to reproduce the trajectory without separating the air outlets (Model A). Then, to further improve the accuracy, we used a separated boundary outlet (Model B).

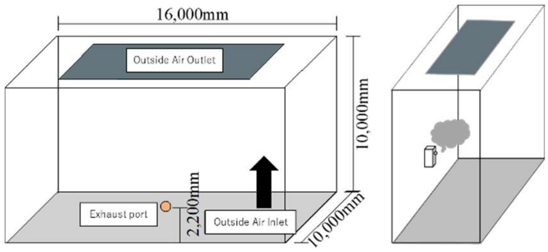


Figure9. Analysis area model

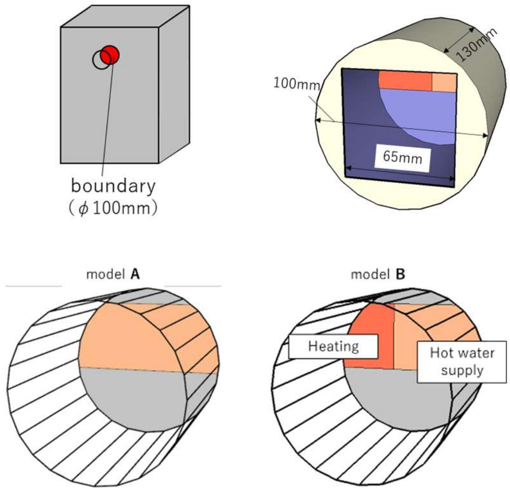


Figure10. Exhaust outlet model

The analysis model and outside air inflow conditions are shown below.

Table2. Analysis conditions

Analysis conditions		
Analytical method		steady-state analysis
Turbulence model		64989
Analysis area		12m × 16m × 10m
Boundary condition	outlet	Wind speed, temperature, CO2 concentration based on actual measurement
	Outside air outflow surface	Pressure outlet
Difference scheme		Secondary accuracy upwind

Table3. boundary conditions (outside air inflow surface)

Boundary condition		
Outside air inflow surface	Velocity[m/s]	0.05
	Temperature[K]	287.69
	Concentration[ppm]	480

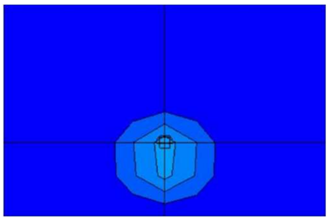
4. Considerations for Improving Reproducibility

4.1 Comparison of the two kinds of velocity

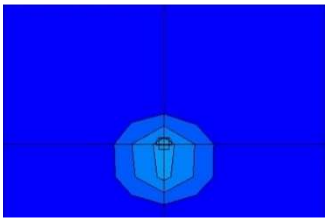
Compare the analysis results using two speeds. One(1) is the velocity calculated with measured CO2 concentration and gas consumption, and the other(2) is the numerical value measured by an anemometer inside the exhaust port.

Table4. boundary conditions

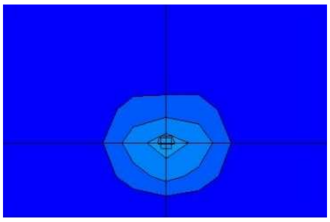
Boundary condition	(1)Calculated by gas consumption	(2)measured
Velocity[m/s]	8.50	8.76
Temperature[K]	338.49	
Concentration[ppm]	64989	



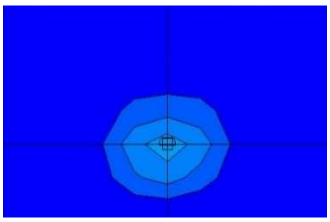
(1) 1.0m from the equipment



(2) 1.0m from the equipment



(1) 2.0m from the equipment



(2) 1.0m from the equipment

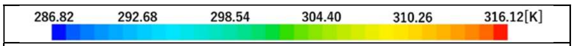


Figure11. "Comparison of the two kinds of velocity"
the temperature distribution

Comparing the results of the analysis, there was little difference in terms of distribution. In terms of core temperature, the diffusivity in (1) was close to the actual measurement results, especially after 2.0 m. Therefore, the wind speed obtained from CO2 is used from now on.

3.2 Correction of orbit centre position

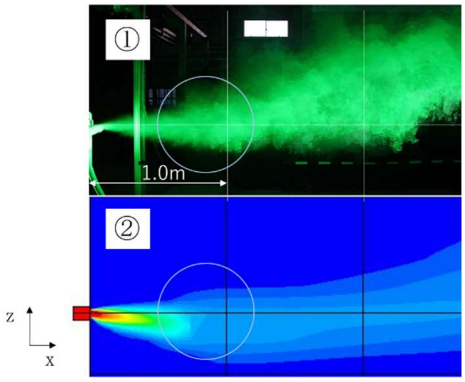


Figure 12. difference in center height of the exhaust orbit(①: results of visualization experiments, ②：analysis results)

From the figure14, it can be seen that there is a shift in the center height of the exhaust track. Both the visualization and analysis results show a gradual rise after sinking downward immediately after exhaust. However, the analysis results showed a stronger downward trend than actual. As for the cause of the downward trend, we found that the exhaust jet is hitting the wall above the model.

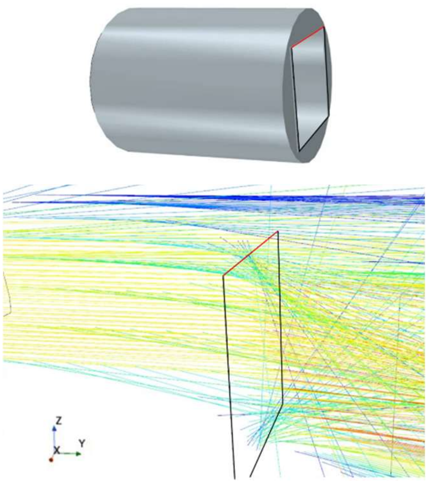
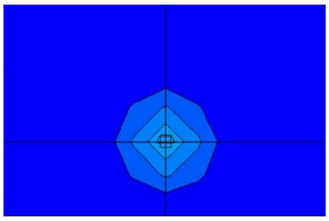


Figure13. Velocity vector diagram near exhaust model

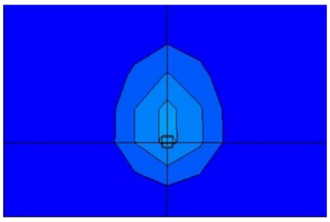
As an improvement measure, apply a correction of 15% of the main jet velocity upward to the boundary surface.

Table5. boundary conditions

Boundary condition	
Velocity(x,y,z)[m/s]	8.50 , 0 , -1.275
Temperature[K]	338.49
Concentration[ppm]	64989



At 1.0m from the equipment



At 2.0m from the equipment

Figure14. “Correction of orbit centre position” the temperature distribution

At 1.0m from the equipment, the orbit height of the exhaust was close to the actual measurement results. And at 2.0m, the results showed the same rise as in reality. It found that adding z-direction correction to the boundary surface is effective in adjusting the orbit height.

3.3 reproducing of deflection

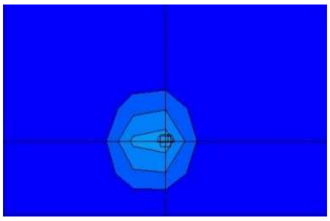
Reproduce the inside of a device (Figure9), and set the conditions by systematically dividing the blowing boundary surface. For each blowing condition, we used the temperature and CO2 concentration proportional to the gas consumption, assuming the same speed.

Table6.heating and hot water supply

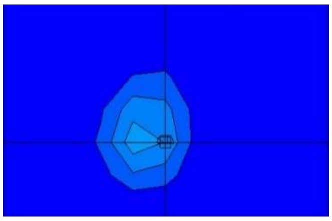
Boundary condition	Hot water supply	heating
Velocity[m/s]	8.50	
Temperature[K]	342.05	332.03
Concentration[ppm]	69260	57235

No deflection appeared as in the actual measurement results by dividing boundary surface, and the exhaust went straight.

So, to reproduce the deflection, we applied a correction of 15% of the velocity towards the weaker output (from hot water supply to heating) and got the following results.



At 1.0m from the equipment



At 2.0m from the equipment

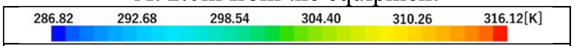


Figure15. “Reproducing of deflection” the temperature distribution

The distribution is now closer to the actual orbit.

5. Conclusion

In the Free-flowing state, deflections to the left or right were observed due to the difference in power output, which may be attributed to the blowing shape inside the device.

Four types of boundary condition setting methods were examined. At the results, we found that using the velocity calculated from the gas consumption is more suitable for reproducing the diffusion state close to the actual measurement. In addition, adding a correction of 15% of the velocity in the +Z and +Y directions is effective in adjusting the orbit position.

6. References

1)Nishizawa et al., Study on installation of ventilation opening in common corridor apartment houses where gas heaters are installed, ROOMVENT conference 2020

The datasets generated during and/or analysed during the current study are not available because it is currently in preparation, but the authors will make every reasonable effort to publish them in near future.