Prediction of Plume above Residential Cooking Range by means of CFD analysis

Yuji TSUMURA[1], Hisashi KOTANI[1], Toshio YAMANAKA[1],

Kazunobu SAGARA [1] and Yoshihisa MOMOI [1]

[1] Dept. of Architectural Engineering, Osaka University, Suita 565-0871 Osaka, Japan

Abstract

Recently, Island kitchen is installed in many houses in Japan. In the island kitchen, the cooking range is placed on the island unit, and the center hood is located above the cooking f low in the room. In such a kitchen environment, there is a possibility that cross winds or other disturbing flow may reach the cooking zone. For this reason, pollutant fumes from the cooking range may be diffused to residential zone. CFD analysis is an effective measure for comprehending such a diffusion of the thermal plume. In comprehending the diffusion of the thermal plume above the cooking range by CFD analysis, a variety of plume models are prop osed as boundary conditions. In this study, "Pot-side wall jet model" is proposed. The velocity and temperature around the pot side are measured, and the profiles are regressed t o formula of two-dimensional jet flow. This means that the thermal plume with high temperature from the gas cooking range is modeled. This paper aims to demonstrate the usefulness of Pot-side wall jet model in analyzing the thermal plume in the large laborato ry where a gas cooking range with a pot is only installed. In this paper, the results of CFD analysis are compared with the experimental results.

Keywords: CFD analysis, Residential Cooking Range, Thermal Plume

Introduction

In order to predict the diffusion of the thermal plume by CFD analysis, the accuracy of the simulated thermal plume is very important. In predicting the plume, a variety of plume models are used as boundary conditions [1] [2] [3]. In this study, “Pot-side wall jet model” is proposed, where the velocity and temperature profiles are approximated by experimental formula of two-dimensional jet flow. The main purpose of this paper is to demonstrate the accuracy and the usefulness of Pot-side wall jet model in analyzing the island kitchen environment. In this paper, three modeling methods are applied for predicting the thermal plume above the range in CFD analysis. One is “Pot-side wall jet model” (hereinafter called Pot-side model), another is “Velocity fixed plume model” (Velocity model) and the other is “Heat generation model” (Heating model).

In Pot-side model, velocity and temperature around the pot are fixed. This means that the thermal plume with high temperature from the cooking gas range is modeled. On the other hand, in Velocity model, the heat generation rates are given on the side of pot, and the velocity at the height of 100mm from the top of pot is fixed. In Heating model, thermal plume is modeled simply by the heat generation rates given on the side of pot.

CFD analysis is designed to compare the experimental results. The velocity and temperature profiles from a gas cooking range are measured at the height of 100mm, 200mm, 300mm, 500mm, and 700mm above the top of the pot .The details can be found in the previous study [4]. The analyzed results by three modeling methods are compared with the measured velocity and temperature of the thermal plume above the pot.

2. Method

2.1 CFD Analysis

CFD analysis is designed to be compared with the experimental results. In order to figure out the characteristics of thermal plume, the measurements were conducted in a large laboratory where no range hood was equipped [4]. Figure 1 shows the calculated domain for CFD analysis. The calculated domain is based on the shapes and dimensions of the laboratory. Table 1 shows summary of CFD analysis. Boundary conditions for the laboratory are shown in table 2. The magnitude of the velocity of supply air was well-argued to avoid both the backflow at the ceiling and the disturbance beside the range. Boundary conditions around pot in each modeling method are described in the following chapters.

CFD analysis and experiments are conducted on the pot of 200mm diameter in boiling situation with the gas range, whose heat inputs are 4.2kW and 1.13kW.

2.2 Pot-side Wall Jet Model

Velocity and temperature around pot were measured by Laser Doppler Velocimetry (LDV) and thermocouple [4]. The measured velocity and temperature are approximated by a regression formula of two-dimensional wall jet, which is proposed by Tamura and Kotani. In Equation (1), vertical component of velocity and temperature rise are presented as a function of the horizontal distance from wall.

$$\frac{\phi}{\phi_m} = 1.479\eta^{1/7}\left[1 - erf\left(0.6776\eta\right)\right] \quad \cdot\cdot\cdot \text{(Eq.1)}$$

$$\left[\begin{array}{ll} \phi = u \text{ (Velocity [m/s]),} & \eta = x / b_{1/2} \\ \quad \text{or } t \text{ (Temperature rise [℃] ※1[mm]} \leqq x) & b = \text{Characteristic length} \\ \text{error function: } erf(x) = \frac{2}{\sqrt{\pi}}\int_0^x e^{-t^2}dt & \end{array}\right]$$

The form of the Equation is proposed by Verhoff cited in the textbook of mechanical engineering [5]. In addition, Φ_m is the maximum value of the velocity and temperature. Half maximum of full width $b_{1/2}$ is calculated by least squares. The conceptual diagram of two-dimensional wall jet is shown in Figure 2. Parameters used in modeling are shown in Table 3.

In this model, pot-side area where velocity and temperature were measured is defined to be out of the analysis domain (This pot-side area is termed “Pot-side Zone”). The upper surface of Pot-side Zone is defined as an inlet condition and the lower surface is defined as an outlet condition (Figure 3). In this model, the heat transfer by radiation is not considered (Table 4). Because its heat generation is considered to be determined by the velocity and temperature at the water surface of the pot and the upper surface of Pot-side Zone. Boundary conditions around pot are shown in Table 5.

2.3 Velocity Fixed Plume Model

This model follows the modeling method proposed by Kondo et al. [2]. Figure 4 and Table 6 show the detail and boundary conditions around the pot, respectively. The heat generation rates are given on the side of pot. And axial velocity at the height of 100mm from the top of pot is fixed. The fixed value is decided by approximating the measured results by a regression formula. Table 7 shows the coefficient of the formula. In addition, radiation is taken into account in this model. Therefore, the temperature at the surface of pot is adjusted to be around 100℃ by means of fixing the heat transfer coefficient at the side of pot. The heat transfer coefficients in the case of heat input of 4.2kW and 1.13kW are determined as 10000 $W/(m^2K)$ and 1000 $W/(m^2K)$, respectively. In this model, heat transfer by radiation is considered.

2.4 Heat Generation Model

Figure 5 and Table 8 show the detail and boundary conditions around the pot in Heat generation model. It is well known that using this kind of modeling methods may underestimate the diffusion of the plume and the width of plume smaller. This model follows the modeling method applied in the previous study [6]. In this model, heat transfer by radiation is not considered.

3. Results and Discussion

Profiles of vertical component of velocity and temperature rise at the each level are shown in Figure 6 and Figure 7.

3.1 Experimental Results

It should be noticed that the maximum temperature rise becomes lower as the measurement height increases. In contradiction to temperature, the maximum velocity becomes higher as the height increases from 100mm to 700mm.

3.2 CFD Analysis in Case of 4.2kW

In Heating model, the maximum magnitudes of temperature and velocity are higher than experimental results. And the width of plume is smaller than experimental results. In Velocity model, it is observed that difference in velocity between CFD and experiment becomes larger as the level becomes higher. This difference seems to come from the disagreement of temperature rise. Temperature above pot is higher than experimental result. Temperature around pot has a great influence in the development of thermal plume. In Pot-side model, both velocity and temperature are in better agreement with experimental results than those in other two models.

Figure 8 shows the distribution of temperature in each model in the case of heat input of 4.2kW. In Pot-side model, it is observed that the width of plume is larger and temperature is lower than those in other models. Besides, in Velocity model, it is observed that temperature around range is a little high. The reason is thought that heat transfer by radiation is considered in Velocity model.

3.3 CFD Analysis in Case of 1.13kW

The profiles of each model have similar characteristics to those in the case of heat input of 4.2kW (Figure 7).

4. Conclusion

The accuracy of the simulated thermal plume is very important in predicting the diffusion of the thermal plumes above the cooking gas range by CFD analysis. It is observed that both velocity and temperature in applying Pot-side rising air model are in better agreement with experimental results than those in other models. So, it can be said that Pot-side rising air model is a higher accuracy modeling method. It also can be said that Pot-side rising air model is a versatile model because it is more suitable than other models when cross winds or other disturbances reach the cooking zone.

After this study, we are scheduled to improve the accuracy of Pot-side rising air model and expand its versatility.

5. Acknowledgements

The authors are grateful to Ms. Asako Tamura who is the former graduate student of Osaka University and Osaka Gas Co., Ltd for supporting the part of this research.

6. Reference

1. Momose T., Yamanaka T., Kotani H., and Yamamoto A. "Modeling and Measurements of Plume above Commercial Cooking Stoves", *J. Environ. Eng., AIJ*, No.567, pp.49-56, 2003 (In Japanese)

2. Kondo Y., Abe A., Miyafuji A., Aizawa Y., and Akagi K. "CFD Modeling of Thermal Plume over a Cooking Pot and Experiments under Heating Condition", *J. Environ. Eng., AIJ*, Vol.73 No.634, pp.1383-1390, 2008 (In Japanese)

3. Sakaguchi J., Akabayashi S., Tomioka S., and Tomaru E. "Study on the thermal environment and air flow distribution in the house kitchen by CFD analysis: Part 1 Study on the temperature, air flow and contaminant distribution above the cooking heater" *Summaries of technical papers of Annual Meeting of AIJ*. D-2: pp.673-674, 2008 (In Japanese)

4. Tamura A., Kotani H., Sagara K., Yamanaka T., and Momoi Y. "Capture Efficiency of Ceiling-mounted Hood in Island Kitchen: Part 3 Simulation Method of Plume from Residential Kitchen Range by CFD analysis" *Summaries of technical papers of Annual Meeting of SHASE Kinki Chapter*., pp.5-8, 2009 (In Japanese)

5. Shakochi T. "Jet Flow Engineering – Fundamentals and Application" *Morikita Publishing Company*, 2004 (In Japanese)

6. Tamura A., Sagara K., Yamanaka T., and Kotani H. "Ventilation Performance of Residential Kitchen Range Hood –Capture Efficiency of Island Hood", *Proceedings of AIVC 29th Conference, Kyoto Japan*, pp.14-16, 2008

Table 1

Summary of CFD Analysis

CFD Code	FLUENT6.3
Turbulence Model	Standaard k-εModel
Density	Imcompressive-ideal gas
Algorism	Steady State (SIMPLEC)
Discretization Scheme for Advection Term	Second Order Upwind

Table 2

Boundary Conditions for Laboratory

Wall Condition		General Log-Low, Adiabatic
Inlet	Supply -Opening	0.02[m/s], 18[℃] k=0 [m^2/s^2], ε=0 [m^2/s^3]
Outlet	Ceiling	Pressure : 0 [Pa]
		(For Backflow: 18[℃], k=0 [m^2/s^2], ε=0[m^2/s^3])

Table 3
Parameters for Modeling
in Pot-side model

	temperature (t)		velocity (u)	
	t_m [℃]	$b_{1/2}$ [mm]	u_m [m/s]	$b_{1/2}$ [mm]
4.2 kW	372.9	0.022	0.95	0.022
1.13kW	88.5	0.011	0.44	0.014

Table 4
Consideration of Heat Transfer
by Radiation

Pot-side model	unconsidered
Velocity model	Discrete Ordinates Method
Heating model	unconsidered

Table 5 Boundary Conditions around Pot
in Pot-side model

		Heat Input	
		4.2[kW]	1.13[kW]
Inlet	Water -Surface	0.0408[m/s]	0.0136[m/s]
		Temperature: 100[℃] (※material: Water Vaper) $k=6.00\times10^{-6}$ [m^2/s^2], $\varepsilon=1.72\times10^{-7}$ [m^2/s^3]	
	Upper Surface of Pot-side Zone	Teperature and Velocity are calcurated by an experimental formula of two-dimensional wall jet (Eq. 1)	
Outlet	Lower Surface of Pot-side Zone	Magnitude of Velocity is equivalent at Upper Surface of Pot-side Zone	

Table 6 Boundary Conditions around Pot
in Velocity model

		Heat Input	
		4.2[kW]	1.13[kW]
Inlet	Water -Surface	0.0408[m/s]	0.0136[m/s]
		Temperature: 100[℃] (※material: Water Vaper) $k=6.00\times10^{-6}$ [m^2/s^2], $\varepsilon=1.72\times10^{-7}$ [m^2/s^3]	
Heat Flux	Pot Side	66182[W/m^2]	14908[W/m^2]
		Area of Heat genarating surface: 3.77×10^{-2} [m^2]	

Table 7 Coefficient of Regression Function in Heating model

($y=a+b|x|+c|x|^2+d|x|^3+e|x|^4+f|x|^5$ ※x: horizontal distance from the center of pot [mm])

Heat Input	Coefficient					
[kW]	a	b	c	d	e	f
4.2	4.982×10^{-1}	-1.546×10^{-2}	1.015×10^{-3}	-1.564×10^{-4}	8.906×10^{-8}	-1.740×10^{-10}
1.13	3.569×10^{-1}	-6.597×10^{-3}	4.875×10^{-4}	-1.005×10^{-5}	7.413×10^{-8}	-1.836×10^{-10}

Table 8 Boundary Conditions around Pot
in Heating model

		Heat Input	
		4.2[kW]	1.13[kW]
Heat Flux	Pot Side	49632[W/m^2]	11181[W/m^2]
		Area of Heat genarating surface: 5.03×10^{-2} [m^2]	

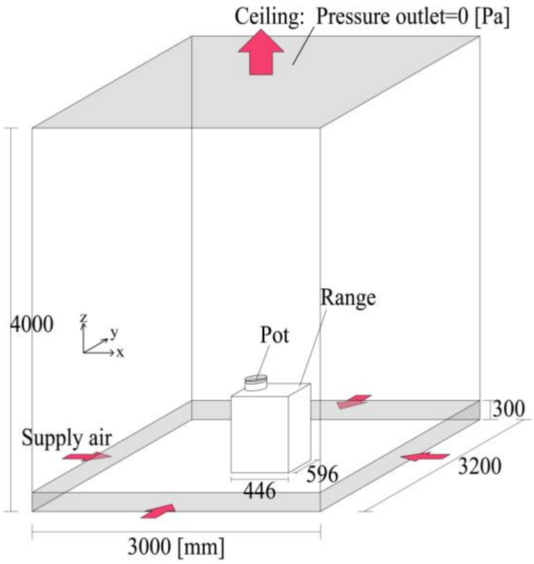


Fig. 1 Calculated Domain

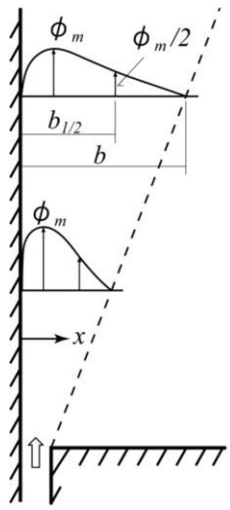


Fig. 2
Flow Model of Two Dimensional Wall Jet

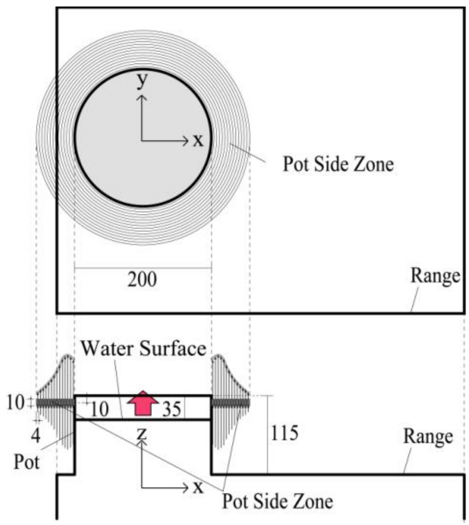


Fig. 3 Detail around Pot in Pot-side model

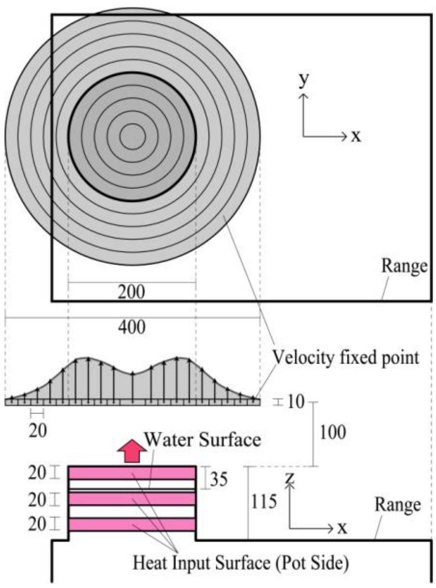


Fig. 4 Detail around Pot in Velocity model

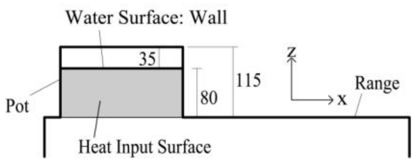
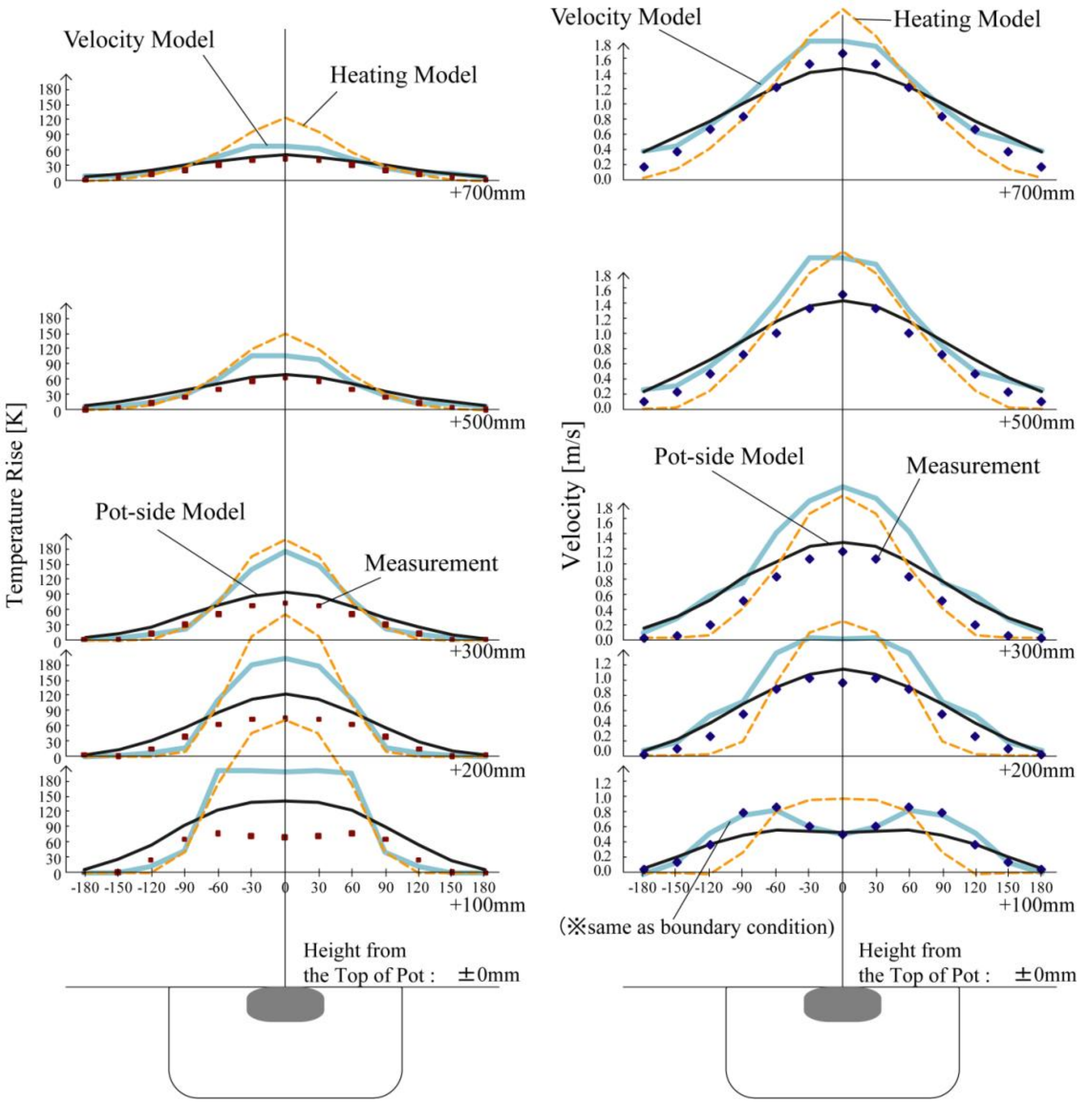
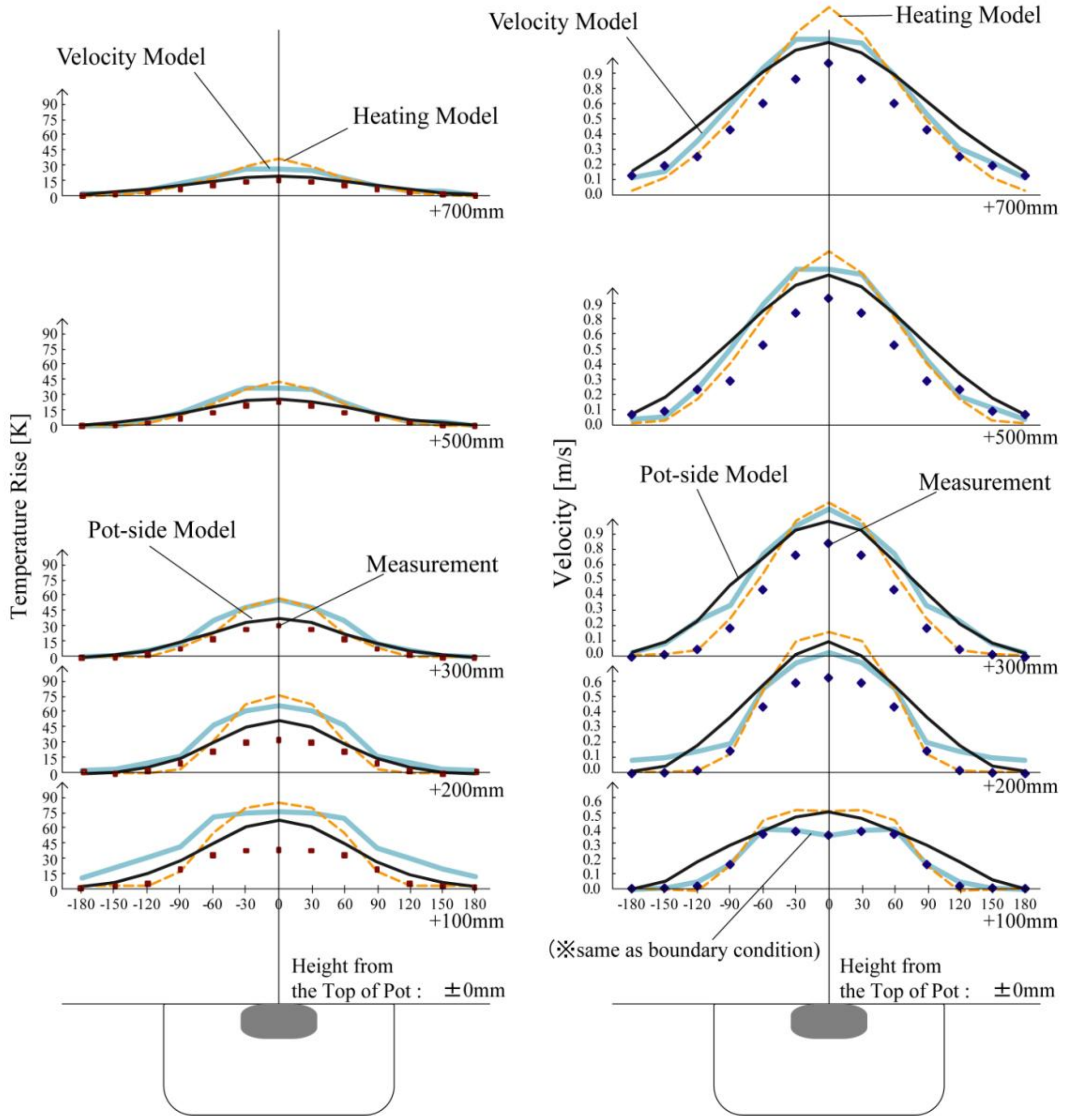


Fig. 5 Detail around Pot in Heating model



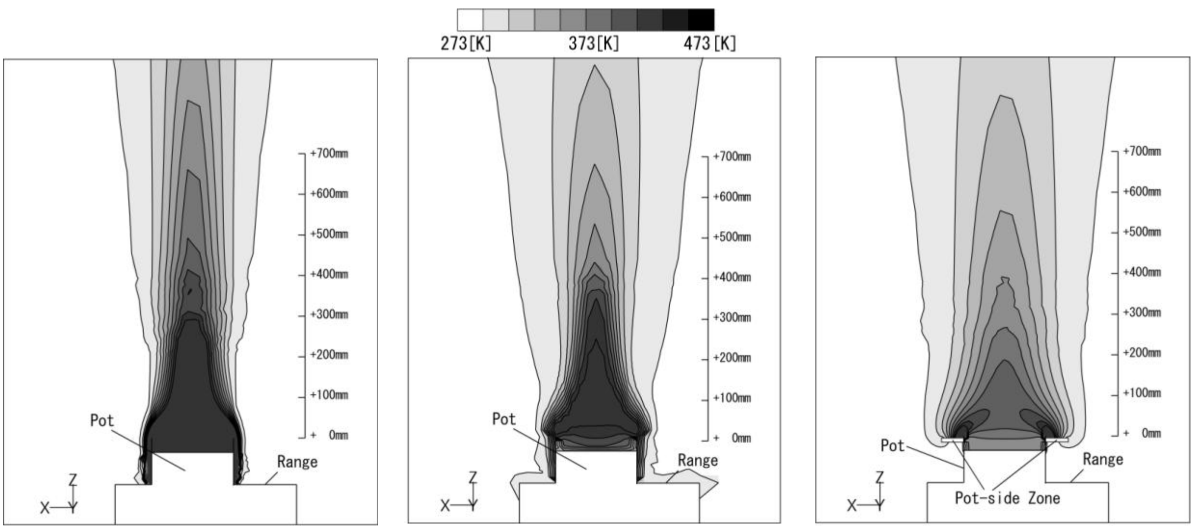
a) Temperature Rise **b)** Velocity

Fig. 6 Distributions of Temperature and Velocity (4.2kW)



a) Temperature Rise **b)** Velocity

Fig. 7 Distributions of Temperature and Velocity (1.13kW)



a) Heating Model **b)** Velocity Model **c)** Pot-side Model

Fig. 8 Temperature Distribution on Vertical Section (4.2kW)