Proposal of a design method for radiant ceiling cooling system using CFD analysis

Rio Matsumoto[1], Jianan Liu[1], Yasuyuki Shiraishi[1], Fujio Tamura[2], Daiki Yamashina[2], Masahiro Yamamoto[2] and Sae Senda[2]

1 The University of Kitakyushu
1-1 Hibikino, Wakamatsu, Kitakyushu, Fukuoka
808-0135, Japan

Email：f4mbb023@eng.kitakyu-u.ac.jp

2 AXS Satow Inc.,
AXS Building, 2-10-12 Yokoami, Sumida, Tokyo.
130-0015, Japan

ABSTRACT

In recent years, the adoption of water-based radiant ceiling cooling systems has been increasing in Japan with the aim of realizing comfort and energy savings. Conventionally, when designing radiant cooling systems, the target operative temperature for the indoor thermal environment is set, but these are usually combined with convection air conditioning system, which do not always achieve the target value during summer cooling. In addition, several parameters must be assumed at the design stage, originally not only operative temperature, but also temperatures of uncooled surface and radiant panel surface, etc. Therefore, it may not be possible to achieve the expected indoor thermal environment under the design conditions, which may affect the estimation of the cooling capacity of radiant panels and the calculation of the number of panels required.

This study proposes a new design method that incorporates CFD analysis based on the conventional design method suggested by ASHRAE. By incorporating CFD analysis, several parameters can be determined at the same time under the assumed environment. In addition, determination of the optimal cooling capacity of radiant panels can be used to realize the target operative temperature by feedback control utilizing the results of CFD analysis. Therefore, in this study, the proposed method was used to redesign a real office with a radiant ceiling cooling system, and the effectiveness of the proposed method was evaluated. The results showed that the cooling capacity of radiant panels can be updated and that the number of panels required can be reduced without compromising indoor thermal comfort.

KEYWORDS

Radiant Ceiling Cooling System, Design Method, CFD Analysis, Operative Temperature

1 INTRODUCTION

In recent years, radiant cooling systems [1][2] have been increasingly adopted for offices and other applications because of their high comfort and energy-saving performance. Usually, panel cooling capacity diagrams are used in the design of radiant cooling systems, and it is necessary to set the target operative temperature (OT) for the indoor thermal environment. However, because radiant cooling systems are combined with convection air conditioning systems with the aim of preventing condensation on panel surfaces, especially during summer cooling, it is not always possible to achieve the target OT. Air temperature is also subject to the influence of the surrounding environment, so the processing heat load due to ventilation and convection air conditioning systems also includes a certain error, i.e., difference from expected value.

To solve the above problems, this study sought to establish an elaborate method for designing a radiant ceiling cooling system utilizing Computational Fluid Dynamics (CFD) analysis. This paper proposes a design method and investigates its effectiveness, using actual summer measurement data of an office building with a radiant ceiling cooling system.

2 A DESIGN METHOD USING CFD ANALYSIS

We propose a design method that incorporates feedback control using CFD analysis based on the conventional design method proposed by American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE) [3], which assumes the influence of the cooling capacity of radiant panels on the indoor thermal environment and its use with convection air conditioning system.

Figure 1 shows the proposed design flow of the radiant ceiling cooling system. The two new steps added to the conventional design method are marked in red boxes. In conventional design method, OT is calculated and evaluated in a later process, but this design method assumes steady state conditions and determines the assumed operating temperature (OT^*), assuming air temperature = OT. In this study, OT is calculated as the average of air temperature and mean radiation temperature (MRT). Next, based on the indoor heat load and required ventilation volume, the processing heat load (total heat) by convection air conditioning and radiant cooling systems (sensible heat) are calculated. Determine the cooling capacity of radiant panels based on the temporary processing heat load by radiant cooling and determine the required panel area. Conventional design methods then determine minimum permissible effective cooling panel surface temperature and the area-weighted average uncooled surface temperature (AUST). The calculation of AUST is based on the simplified assumption that temperature of uncooled surface is air temperature. AUST is used to calculate MRT and even OT to assess the indoor thermal environment and determine panel area. Whereas this design method incorporates CFD analysis to predict the indoor thermal environment distribution that reproduces the real phenomena in detail. Specifically, it is possible to calculate and determine several parameters at once, which originally had to be assumed at the design stage, such as the cooling capacity of radiant panels and temperature of radiant panels surface, reflecting the effect of temperature of uncooled surface and convection air conditioning system. Therefore, the method is expected to improve the design errors that occur in conventional design methods. The results of the comparison of each parameter with conventional methods are presented in Chapter 4.

Furthermore, after determining the outline of convection air conditioning and radiant cooling systems, the layout of radiant panels is determined and CFD analysis is performed in the verification assuming steady-state conditions. From the results of the CFD analysis, the analytical value (OT_{ci}) is compared with the assumed operative temperature (OT^*), and the cooling capacity of radiant panels is revised and updated until the assumed value and the analytical value are almost in agreement. Finally, based on the determined the cooling capacity of radiant panels, the water supply temperature is determined using a panel cooling capacity diagram, and the design flow is completed.

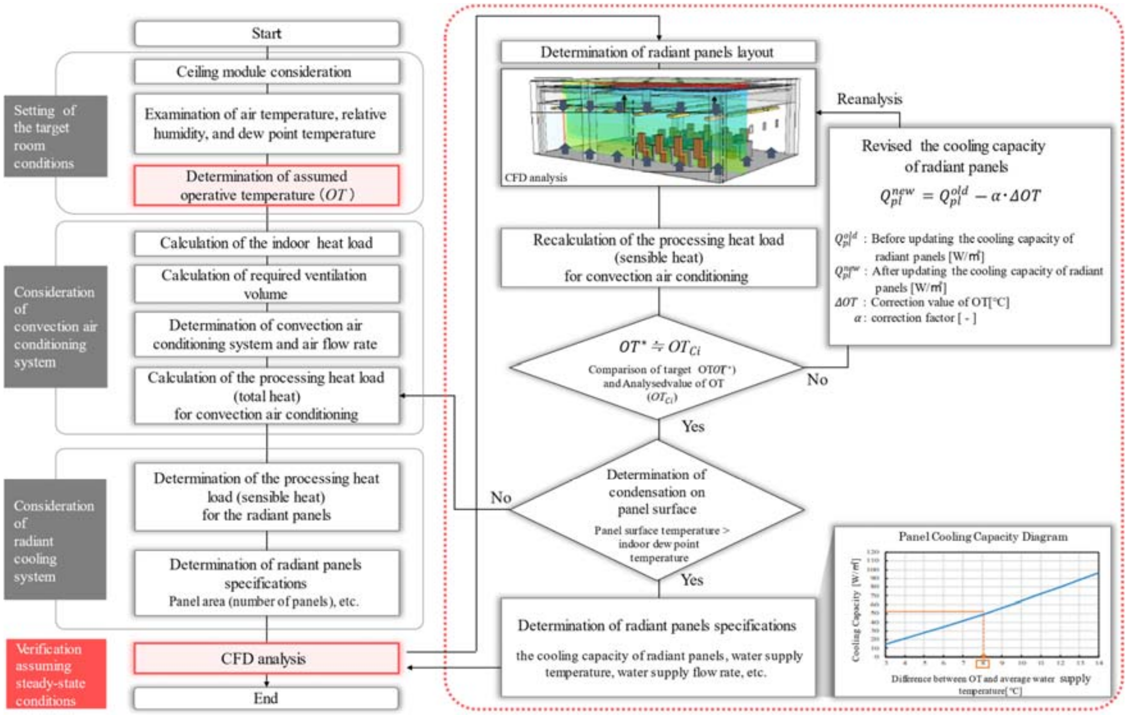

Figure 1: The proposed design flow of the radiant ceiling cooling system

3 OVERVIEW OF THE SUBJECT FACILITY AND CFD ANALYSIS MODEL

3.1 Subject facility

The building is a city hall completed in June 2022 (Fig. 2). This study focuses on an area of the third floor used as office space, where a radiant ceiling cooling system is installed (Fig. 3).

Figure 2: Building exterior

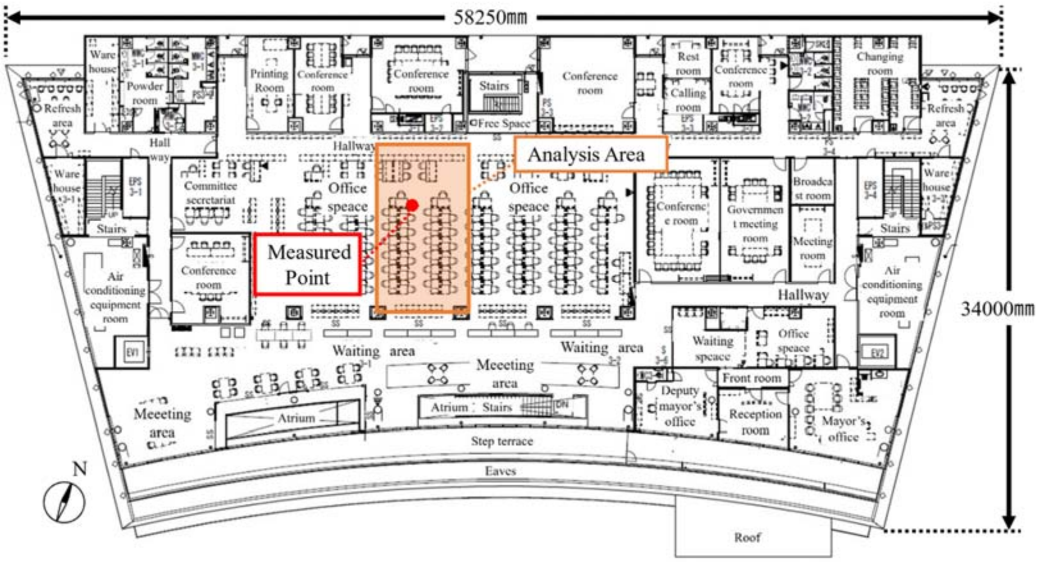

Figure 3: Floor plan

3.2 CFD analysis model and analysis conditions

The analysis model of the target area is shown in Fig. 4, the analysis model of the radiant panels in Fig. 5 and the analysis conditions in Table 1. Sixty-eight ceiling radiation panels (total area: 48.96 m^2, ceiling coverage ratio: 53%) were installed in the area analysed. During office hours, outside air is supplied at a constant temperature and flow rate from three air outlets (supply outdoor air：SOA). Return air (RA) is also taken in at a constant flow rate at the return air outlet. The difference between SOA and RA is exhaust (as exhaust air: EA) in the toilet via the common space north of the office.

The modelling of radiant panels was simplified to reduce the computational load of the analysis. The chilled water pipes were changed to 1 mm panels with the physical properties of water, and the processing heat load of the chilled water was given as the analysis condition for that panel. The thickness of the aluminium panel was set to 10 mm. However, the thermal capacity of the panel was set to be the same as that of a 1 mm thick panel, and the thermal conductivity in the thickness direction was set so that the thermal resistance value of the panel in the thickness direction was set to be the same as that of 1 mm thick aluminium.

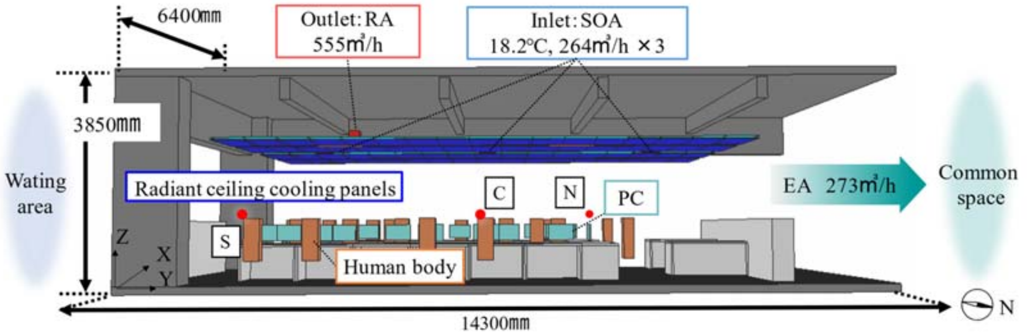

Figure 4: Analysis model

Figure 5: Radiant ceiling cooling panels model

Table 1: Analysis conditions for CFD analysis

Domain	6.4m(X)×14.3m(Y)×4.15m(Z)	
Mesh	125(X)×279(Y)×81(Z)= 2,824,875	
Inlet boundary conditions	[Present System]	Temperature: 18.2□, Velocity: 264*3 m^3/h $k_{in1}=(U_{in}/10)^2, \varepsilon_{in1}=C_\mu^{3/4}\cdot k_{in1}^{3/2}/l_{in1}$
	[Case 1]	Temperature: 18.2□, Velocity: 264*3 m^3/h $k_{in2}=(U_{in}/10)^2, \varepsilon_{in2}=C_\mu^{3/4}\cdot k_{in2}^{3/2}/l_{in2}$
	[Case 2]	Temperature: 20.0□, Velocity: 200*3 m^3/h $k_{in3}=(U_{in}/10)^2, \varepsilon_{in3}=C_\mu^{3/4}\cdot k_{in3}^{3/2}/l_{in3}$
Outlet boundary conditions	RA	Velocity: 555m^3/h
	North Side	[Present System] 237m^3/h
		[Case 1] 237m^3/h
		[Case 2] 55m^3/h
Turbulence model	Linear Low Reynolds turbulence model	
Advection scheme	QUICK	
Wall boundary conditions	Velocity	Analytical wall function
	Temperature	Logarithmic law
Indoor heat load	Human (20 person): 69W/person, Lighting (2 lights): 398W/unit, PC (20 units): 75W/unit	

U_{in1}, U_{in2} , U_{in3}: Outlet air wind speed [m/s], k_{in1} ,k_{in2} ,k_{in3}: Outlet air turbulence energy [m^2/s^2], $\varepsilon_{in1}, \varepsilon_{in2}, \varepsilon_{in3}$: Dissipation rate of k_{in} [m^2/s^3], $C_{\mu1}$, $C_{\mu2}$, $C_{\mu3}$: Model constant (=0.09) [-], l_{in1}, l_{in2}, l_{in3}: Length scale [m]

4 OVERVIEW OF THE SUBJECT FACILITY AND CFD ANALYSIS MODEL

4.1 Verification of the prediction accuracy of the analysis model

Figure 6 shows predictive validation results of the analysis model in vertical air temperature distribution. This analysis evaluated the prediction accuracy of the analysis model by comparing the vertical air temperature distribution, using the data of the subject office, which was measured in the summer. The assumed number of occupants was set to 20 based on the actual measurement data. From the results of the prediction accuracy verification, root mean square error (RMSE) of the vertical air temperature distribution was 0.30°C, which indicated sufficient prediction accuracy.

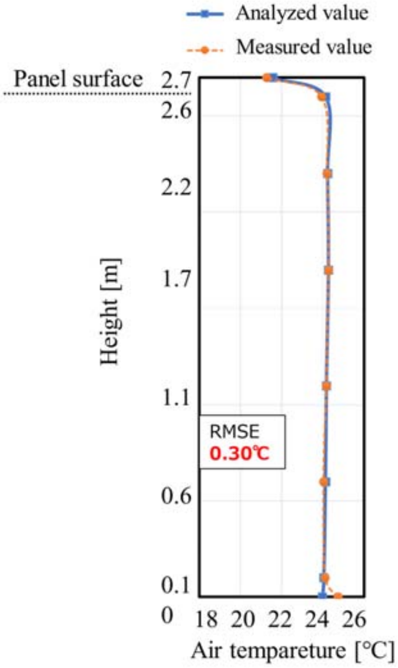

Figure 6: Predictive validation results of the analysis model in vertical air temperature distribution

4.2 Thermal environment evaluation by CFD analysis

The indoor thermal environment was assessed by the occupant area mean OT in the office and the predicted mean vote (PMV). The metabolic rate and the clothing insulation when calculating PMV were 1.2 met and 0.6 clo, respectively, which are common values in summer. The relative humidity was assumed to be 50% because no humidity measurements were performed in this study. PMV was also calculated from the average of three points (S, C, and N) of the room's central sensor (Fig. 4; height, 1.1 m).

The average OT of the workspace was 24.5°C, resulting in a generally uniform the indoor thermal environment. PMV was 0.15, which was confirmed to be within the comfortable range. However, in the present operation, the assumed cooling capacity of radiant panels was only 54% of the rated cooling capacity. Therefore, it was suggested that the number of radiant panels could be reduced, and the processing heat load ratio by radiant cooling could be increased.

4.3 Redesign of radiant cooling system using the proposed method

In this section, case studies on the redesign of radiant cooling system are carried out (Table 2). In Case 1, the present blowout conditions were maintained, and the minimum number of panels (44 panels) were installed. The calculated panel reductions were dummy panels and were evenly distributed within the laying area. In Case 2, the blowout conditions were relaxed to increase the processing heat load ratio by radiant cooling. First, considering comfort, the assumed OT was set at 24.4°C (PMV corresponds to approximately ±0). Next, the assumed cooling capacity of radiant panels determined was input into the CFD analysis as an initial value, and the convergence calculation was started. However, the convergence calculation was terminated because PMV was 0.05, reaching the target value (Table 3).

The heat balance before and after the redesign is shown in Figure 7. Case 1 shows that 24 radiant panels can be reduced while maintaining the indoor thermal environment. In addition, the processing heat load ratio by radiant panels increased by 26% when comparing the current situation with Case 2. Furthermore, a comparison of the initial and final convergence values of Case 1 shows the processing heat load by radiant panels from 1,958 to 1,657W through convergence calculations performed in the CFD analysis. These results suggest that the proposed method may have high design accuracy regarding comfort and energy saving. Comparing the analysis results of the conventional design method and the proposed method in Case 1, the air temperature was updated from 24.4°C to 24.3°C, OT from 23.1°C to 24.4°C, temperature of radiant panels surface from 17.3°C to 19.8°C and AUST from 24.2°C to 24.9°C. Furthermore, it was confirmed that air temperature ≠ AUST.

Table 2: Analysis case with redesign

	Blowoff temperature [□]	**Blowoff flow rate [m³/h]**	**Number of panels**	**The assumed cooling capacity of radiant panels [W/m²]**
Case1	18.2	792	44	52.32
Case2	20.0	600	68	53.32

Table 3: Convergence results of CFD analysis

	The cooling capacity of radiant panels [W/m²]	**Air temperature [□]**	**MRT [□]**	**OT [□]**	**PMV [-]**
Initial Assumed Value	61.84	24.40	24.40	24.40	0
Final Convergence value	52.32	24.31	24.42	24.37	+0.05

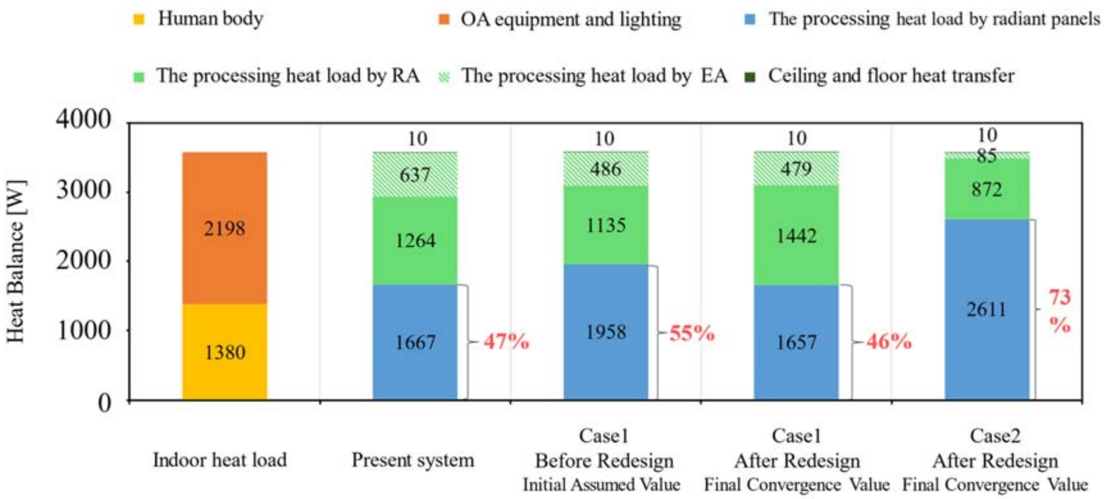

Figure 7: Heat balance before and after convergence calculation

5 CONCLUSIONS

A design method for radiant ceiling cooling system that incorporate CFD analysis into the conventional ASHRAE design method was proposed. Using the proposed design method to redesign an actual building, it was possible to reduce the number of radiant panels without compromising indoor comfort, in addition to improving the processing heat load ratio by radiant cooling. In the future, the effectiveness of the proposed method will be verified by increasing the number of cases in which it is adapted to real properties.

6 REFERENCES

[1] ISO 11855(all parts).(2024).“Building environment design” - Design, dimensioning, installation and control of embedded radiant heating and cooling systems

[2] ISO 18566 (all parts).(2019) “Building environment design” - Design, test methods and control of hydronic radiant heating and cooling panel systems

[3] ASHRAE.(2012) “2012 ASHRAE Handbook HVAC System and Equipment” SI edition-Ch.6 Panel Heating and Cooling, American Society of Heating Refrigerating and Air-Conditioning Engineers