

Khatri, Rahul; Khare, Vaibhav Rai; Kumar, Hemant

Article

Spatial distribution of air temperature and air flow analysis in radiant cooling system using CFD technique

Energy Reports

Provided in Cooperation with:

Elsevier

Suggested Citation: Khatri, Rahul; Khare, Vaibhav Rai; Kumar, Hemant (2020) : Spatial distribution of air temperature and air flow analysis in radiant cooling system using CFD technique, Energy Reports, ISSN 2352-4847, Elsevier, Amsterdam, Vol. 6, Iss. 2, pp. 268-275, <https://doi.org/10.1016/j.egyr.2019.11.073>

This Version is available at:

<https://hdl.handle.net/10419/243888>

Standard-Nutzungsbedingungen:

Die Dokumente auf EconStor dürfen zu eigenen wissenschaftlichen Zwecken und zum Privatgebrauch gespeichert und kopiert werden.

Sie dürfen die Dokumente nicht für öffentliche oder kommerzielle Zwecke vervielfältigen, öffentlich ausstellen, öffentlich zugänglich machen, vertreiben oder anderweitig nutzen.

Sofern die Verfasser die Dokumente unter Open-Content-Lizenzen (insbesondere CC-Lizenzen) zur Verfügung gestellt haben sollten, gelten abweichend von diesen Nutzungsbedingungen die in der dort genannten Lizenz gewährten Nutzungsrechte.

Terms of use:

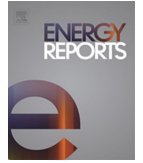
Documents in EconStor may be saved and copied for your personal and scholarly purposes.

You are not to copy documents for public or commercial purposes, to exhibit the documents publicly, to make them publicly available on the internet, or to distribute or otherwise use the documents in public.

If the documents have been made available under an Open Content Licence (especially Creative Commons Licences), you may exercise further usage rights as specified in the indicated licence.



<https://creativecommons.org/licenses/by-nc-nd/4.0/>



The 6th International Conference on Power and Energy Systems Engineering (CPESE 2019),
20–23 September 2019, Okinawa, Japan

Spatial distribution of air temperature and air flow analysis in radiant cooling system using CFD technique

Rahul Khatri^a, Vaibhav Rai Khare^{b,*}, Hemant Kumar^a

^a Manipal University Jaipur, Jaipur 303007, India

^b Malaviya National Institute of Technology, Jaipur, 302021, India

Received 4 October 2019; accepted 22 November 2019

Abstract

This paper describes the air flow analysis and distribution of a radiant cooling system installed in a commercial building. Computational Fluid Dynamics (CFD) simulations using ANSYS were carried out to identify the thermal performance of the radiant cooling system. The calibration of simulated model was done using the actual data and a hypothetical case of conventional cooling system has been evaluated with the help of TRNSYS for comparative analysis. The CFD simulations were carried out to find the effect of the spatial distribution of air for the radiant system and the conventional system. It was recorded that a radiant cooling system maintains uniform air distribution and better average air temperature as compared to a conventional cooling system. Impact of thermal mass on the fluctuations of the temperature was also studied and it was identified that the higher thermal mass prevents high fluctuations in operative temperatures.

© 2019 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Peer-review under responsibility of the scientific committee of the 6th International Conference on Power and Energy Systems Engineering (CPESE 2019).

Keywords: Radiant cooling; TRNSYS; CFD; Building energy simulation; HVAC; Thermal mass

1. Introduction

The radiant cooling systems are gaining interest as an approach for energy-efficiency in buildings for space cooling. As per available literature by Khan et al. [1] and Feustel and Stetiu [2], numerous researches show that radiant cooling system can decrease the energy consumption by about 40% compared to conventional cooling systems. According to Pérez-Lombard et al. [3], Cooling systems in India consumes almost 30%–50% of the total electricity consumption of a commercial building, depending on building type and working schedules. A study conducted by Seo et al. [4] states that the radiant cooling system saves energy by providing water at a lower temperature and using less transport energy, accounting for sensible cooling load only; a separate system should account for the rest of the cooling load.

* Corresponding author.

E-mail address: vrk1007@gmail.com (V.R. Khare).

<https://doi.org/10.1016/j.egy.2019.11.073>

2352-4847/© 2019 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Peer-review under responsibility of the scientific committee of the 6th International Conference on Power and Energy Systems Engineering (CPESE 2019).

A radiant cooling system improves thermal comfort conditions as it handles the load directly and uniformly as stated by Imanari et al. [5]. A study by Tian and Love [6] confirmed that a radiant system perform better compared to a conventional cooling system, subject to the location and climate, system performed best in dry climates.

The difference in the air temperature from head to ankle of a human body could be reduced by increasing the temperature of the supply air. Higher supply air temperature reduces the difference between the air temperatures from head to ankle and achieve better thermal comfort for the occupied condition. It was found by Chiang et al. [7] that if the temperature of supply air is raised to 24 °C from 18 °C, it increases the indoor air temperature by 26.5 °C that reduces the energy consumption. This results in almost 13% energy saving for the chiller.

Conventional systems only take account of mean air temperature (MAT), which causes a rise in load on the system and unequal distribution of air in the zone, which ultimately results in poor thermal comfort conditions for the occupant. While, the radiant cooling system lowers the zone operative temperature by bringing down the MAT of the zone. It also maintains the temperature of the zone throughout the depth or height which provides better thermal comfort conditions for the occupants, without increasing the cooling energy demand.

This study describes the effect of the spatial distribution of air into a radiant cooling system and conventional systems with the help of Computational Fluid Dynamics (CFD). The impact of thermal mass on the performance of radiant cooled buildings is presented and compared in this study. With the dynamic input boundary conditions for surface temperatures and air flow supply provided by TRNSYS Energy Simulation, CFD simulation can effectively predict the dynamic indoor environment.

2. Evaluation approach

An office building located in Hyderabad, India, has been modeled for study. The building has a radiant cooling system that consist of both roof and floor mounted tubular installations. Existing radiant cooling system is combined with Fan Coil Unit (FCU) to cater the latent heat load. The energy model was simulated for energy performance and calibrated with the simulated data. The cooling load for the conventional cooling system was kept same as to compare the results with radiant cooling system. CFD technique was used to study the spatial variation in the flow and thermal properties for specific boundary condition. The CFD model was validated by taking boundary temperatures from the developed energy model of the building. The whole methodology is presented in Fig. 1

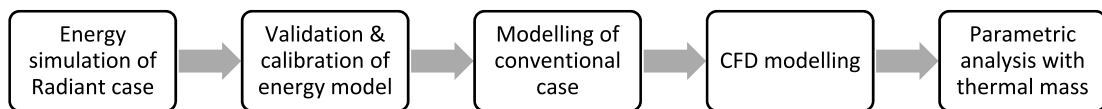


Fig. 1. Methodology of the study.

3. Building modeling

3.1. Building description and input parameters

The modeled building is an office building located in a composite climatic zone in Hyderabad, India. It has approx. 354 m² conditioned space. Fig. 2 shows the 3D model of the building. The energy simulations were run

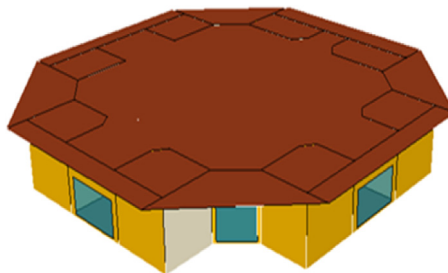


Fig. 2. 3D model of the building.

with a variable pattern of occupancy, and each one was considered to generate 115 W of heat as per ASH [8]. The building envelope and load parameters are shown in Table 1.

Table 1. Building input parameters.

Input parameters	Value
Wall construction	Brick wall (U-value [W/m ² K] = 1.79)
Roof construction	Concrete roof (U-value [W/m ² K] = 1.05)
Window glazing	SHGC = 0.58, U-value = 5.6, VLT = 0.75
Window–wall ratio	8.8%
Plug load	20 [W/m ²]
LPD	8.5 & 20 [W/m ²]

3.2. HVAC system configuration

The HVAC system is coupled with the adjacent building to provide cooling, so the chiller has been auto sized for its given coefficient of performance (COP). A constant-volume FCU system provides ventilation with 20% fresh air only. The zone thermostat controls FCU system operation and radiant temperature schedule determines the response of the radiant system. The fan efficiency was taken 52%, motor efficiency, and pressure rise were taken as 80% and 330 Pa respectively as per Deru et al. [9]. A description of the HVAC system configuration is given in Table 2.

Table 2. HVAC system input parameters.

Parameter	Value
HVAC system type	Radiant system with FCU
Fan design	Constant volume with 0.85 m ³ /s (1800 CFM)
Supply air temp	19–20 °C
Chiller parameters	FCU chiller: 3.1 COP, 12 °C chilled water temperature (leaving) Radiant chiller: 3.5 COP and 16 °C chilled water temperature (leaving)
Radiant pipe details	Diameter = 15 mm, Spacing = 100 mm, Loop length = 4640 m
Zone setpoint temp	24 °C

3.3. Model calibration and validation

Model calibration was done to compare the measured and simulated energy consumption on an hourly basis. It was done by regulating the occupancy and plug load, which have the highest diversity. The measured data included only the cooling energy of the chiller, so this paper discusses only the cooling energy load of the building. Fig. 3 shows the correlation between measured and simulated cooling energy consumption.

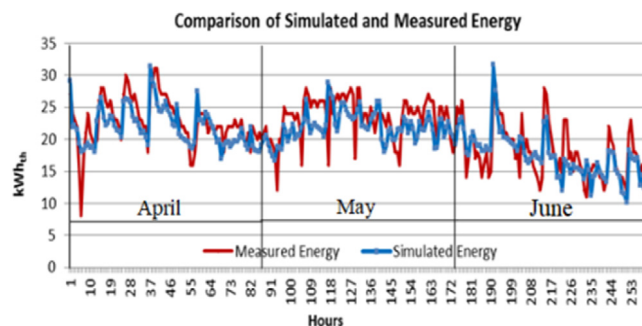


Fig. 3. Comparison of simulated and measured energy.

The calibration results were found to be well within the acceptable limits of the NMBE and CvRMSE criteria as per U.S. Department of Energy [10] which are 2.7% and 13.1% respectively as shown in Table 3; and thus, are adequate.

Table 3. HVAC system input parameters.

Calibration type	Index	Limit		Calibrated model error
		(FEMP)	(IPMVP)	
Hourly	MBE	±10%	–	2.7%
	C _v RMSE	30%	20%	13.1%

4. CFD modeling and simulation

4.1. CFD modeling

In this analysis, ANSYS Workbench was used to prepare the geometry of the domain and the mesh generation. An unsteady state implicit pressure-based solver has been used for numerical simulation using FLUENT. Appropriate boundary conditions were used based on the given condition. The air was assumed to be incompressible ideal gas, and the air flow was assumed to be turbulent. The properties (physical & thermal) of different materials were used, based on type of material. The building geometry and meshed drawing are shown in Fig. 4

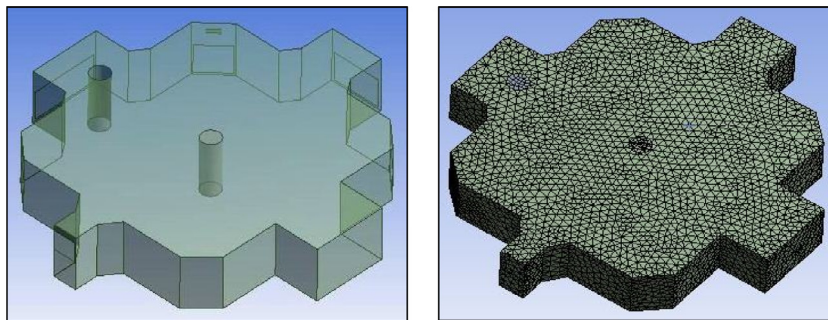


Fig. 4. Geometrical model and meshing of building.

A good mesh quality is essential for performing a valid CFD analysis, therefore after mesh generation, quality of mesh is checked. Grid independence test has also been performed to optimize the mesh size and quality. Table 4 shows the inputs parameter for CFD modeling.

Table 4. Input parameters for CFD modeling.

Parameters	Specifications		
Solver setting	Space	3D	
	Time	Unsteady, First-order implicit	
	Viscous	k-epsilon RNG model	
Solution methods	Radiation	Surface to surface	
	Pressure velocity coupling	Simple	
	Spatial discretization	Pressure	Standard
Solution initialization		Momentum	First order upwind
	Time step size	Energy	First order upwind
	Maximum iteration	0.2 s	25

4.2. CFD model validation

For CFD model validation, hourly readings were taken from the TRNSYS model and applied to both cooling systems. The parameters considered are, supply air temperature, floor temperature, ceiling temperature and the mass flow rate of air for FCUs. The variations in the simulated results showed good agreement for air temperature in both the CFD model and TRNSYS, as shown in Fig. 5. The slight disagreement between the air temperatures is due to its dependency on the load. Thus, this model can be considered as a valid model for further analysis.

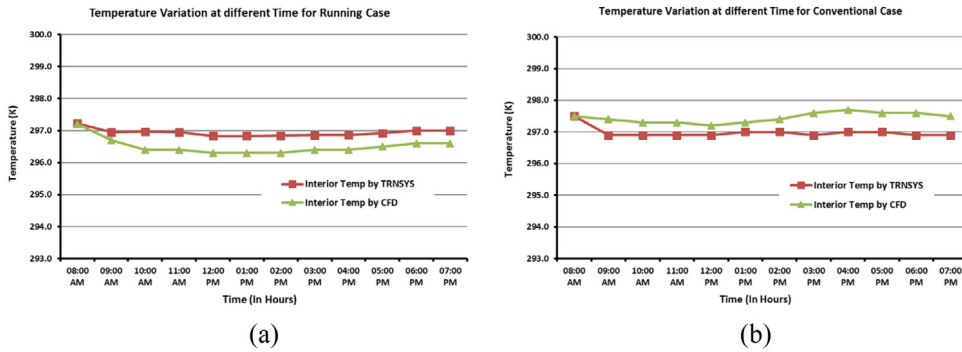


Fig. 5. Temperature variance (a) Radiant case, (b) Conventional case.

5. Results and discussion

5.1. Spatial distribution

The validated CFD model was used to run building simulation for both Radiant Case and Conventional Case in order to analyze the spatial distribution of indoor air temperatures at different room heights and the mean air temperature of the particular zone. The radiant cooling system shows a uniform spatial indoor air temperature with variation in height as shown in Fig. 6. In comparison, the conventional system has a non-uniform variation in temperature. This is mainly due to the installations of FCU’s at a certain height in the conditioned zone.

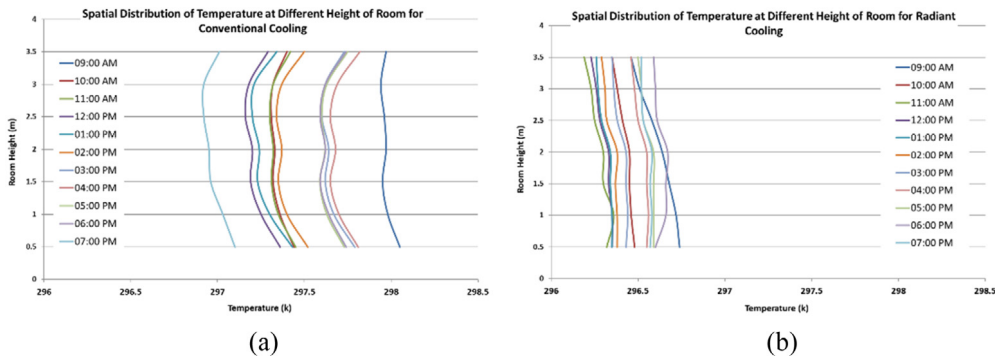


Fig. 6. Spatial air distribution (a) Conventional case (b) Radiant case.

5.2. Mean air temperature

The mean air temperature range for the radiant cooling system and conventional cooling system was calculated by taking average of the zone temperatures at different increments of time. Fig. 7 shows that the radiant cooling system had a higher percentage of time at the mean air temperature and have less fluctuation in mean air temperatures, which will result in better thermal comfort conditions.

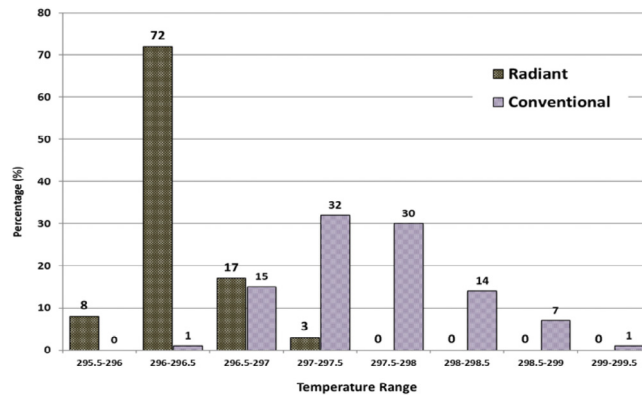


Fig. 7. Average air temperature distribution.

5.3. Temperature contours and velocity vectors

Fig. 8 represents the temperature contour for a given time for both the cases. It shows the gradual cooling of the room at a height of 1.5 m. It can be seen from the contours that after some time, the room temperature became uniform for the radiant case with low temperature variation, while for the conventional case, it shows the non-uniform room temperature with high-temperature variations.

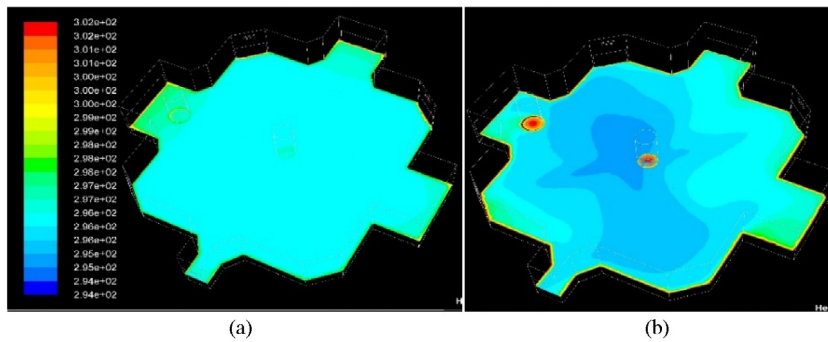


Fig. 8. Temperature contours for (a) Radiant & (b) Conventional case.

In the conventional case, the cold region can be seen at the center of the room. Fig. 9 shows the air velocity vectors for a typical case which shows that the supply air hits the pillar which is in center and thus create the cold region.

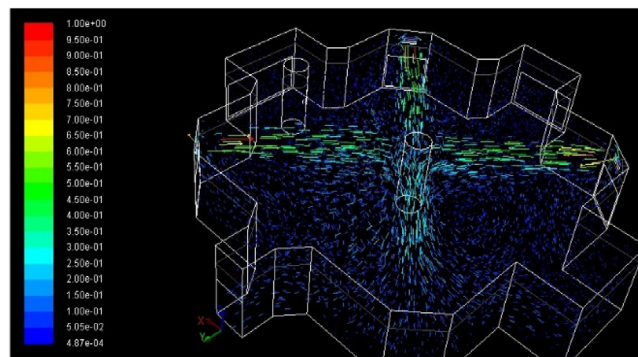


Fig. 9. Air velocity vectors for a typical case.

6. Parametric analysis

It has been found from the previous researches that there is a significant effect of thermal mass on the thermal performance of the radiant cooled building. Four different cases were studied to evaluate the effect of thermal mass on air distribution in the building which are following:

- Case-1: Radiant system OFF before 1-h
- Case-2: Radiant system OFF before 2-h
- Case-3: Radiant system OFF before 3-h; and
- Case-4: Radiant system OFF before 4-h.

In all the above-mentioned cases, the radiant system was shut down at different time interval while the FCU system is scheduled to run continuously. Fig. 10 shows the air distribution of all the above case. The results show that the thermal mass has positive effect on thermal comfort.

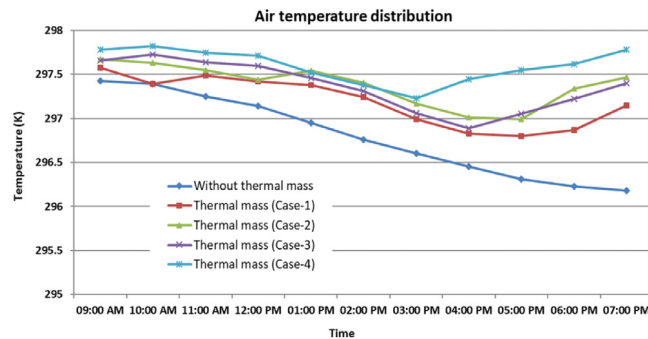


Fig. 10. Air temperature distribution for all cases.

In the developed model with added thermal mass, the fluctuations in the operative temperature are significantly smaller than in the model without thermal mass. Thus, it can be said that the thermal mass plays a significant role for achieving the thermal comfort in conditioned buildings. Therefore, the construction of the building should be done in a manner to achieve a better level of thermal comfort.

7. Conclusion

CFD simulations provide full information about the thermal performance of the building, such as distribution of air temperature, and air flow. This information enables to determine the thermal comfort of a building for the design of comfortable and healthy indoor environments. This study was carried out to identify the thermal performance of a radiant cooling system, for an office building in Hyderabad, India in comparison with a conventional cooling system. An existing cooling setup consisting of an integrated radiant cooling system and FCU's was modeled to assess the thermal performance. The results obtained were validated with the actual energy consumption data. Then, the validated model was used to simulate the thermal consumption of a building using a conventional system to determine the thermal performance for both cases. Spatial air distribution shows that the conventional system have non-uniform distribution of air temperature due to installations of FCU's at a fixed elevation in the conditioned area. Also, the mean air temperature range for both the cases has been computed by taking the average of the zone temperatures at different times. The radiant cooling has higher mean air temperature percentage, which is an important factor for better thermal comfort condition. The effect of thermal mass on the spatial distribution of air temperature of the building was also studied with the developed model. In this analysis, different cases have been modeled to achieve better thermal comfort. The results show that the thermal mass shows positive effect on thermal comfort, in the model with thermal mass, the fluctuations in the operative temperature are significantly smaller than in the model without thermal mass.

References

- [1] Y. Khan, V. Khare, J. Mathur, M. Bhandari, Performance evaluation of radiant cooling system integrated with air system under different operational strategies, *Energy Build* 97 (2015) 118–128.

- [2] H. Feustel, C. Stetiu, Hydronic radiant cooling — preliminary assessment, *Energy Build* 22 (3) (1995) 193–205.
- [3] L. Pérez-Lombard, J. Ortiz, J. Coronel, I. Maestre, A review of HVAC systems requirements in building energy regulations, *Energy Build* 43 (2–3) (2011) 255–268.
- [4] J. Seo, D. Song, K. Lee, Possibility of coupling outdoor air cooling and radiant floor cooling under hot and humid climate conditions, *Energy Build* 81 (2014) 219–226.
- [5] T. Imanari, T. Omori, K. Bogaki, Thermal comfort and energy consumption of the radiant ceiling panel system, *Energy Build* 30 (2) (1999) 167–175.
- [6] Z. Tian, J. Love, Application of radiant cooling in different climate: assessment of office building through simulation. *building simulation 2009*, in: Eleventh international IBPSA conference, 2009, pp. 2220–2227.
- [7] W. Chiang, C. Wang, J. Huang, Evaluation of cooling ceiling and mechanical ventilation systems on thermal comfort using CFD study in an office for subtropical region, *Build Environ* 48 (2012) 113–127.
- [8] ASHRAE Standard 55, Thermal environmental conditions for human occupancy, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta, 2004.
- [9] M. Deru, K. Field, D. Studer, K. Benne, B. Griffith, P. Torcellini, et al., U.S. Department of energy commercial reference building models of the national building stock, National Renewable Energy Laboratory, Colorado, 2011.
- [10] U.S. Department of Energy, M & V guidelines: Measurement and verification for performance-based contracts version 4.0, 2015.