DEVELOPMENT OF PERSONALIZED RADIANT COOLING SYSTEM FOR AN OFFICE ROOM

Vaibhav Rai Khare[1], Anuj Mathur[1], Jyotirmay Mathur[1], Mahabir Bhandari[2]

[1]Centre for Energy & Environment, Malaviya National Institute of Technology, Jaipur, India.

[2]Oak Ridge National Laboratory, Oak Ridge, USA

ABSTRACT:

The building industry nowadays is facing two major challenges – increased concern for energy reduction and growing need for thermal comfort. These challenges have led many researchers to develop Radiant Cooling Systems that show a large potential for energy savings. This study aims to develop a personalized cooling system using the principle of radiant cooling integrated with conventional all-air system to achieve better thermal environment at the workspace. Personalized conditioning aims to create a microclimatic zone around a single workspace. In this way, the energy is deployed only where it is actually needed, and the individual's needs for thermal comfort are fulfilled. To study the effect of air temperature along with air temperature distribution for workspace, air temperature near the vicinity of the occupant has been obtained as a result of Computational Fluid Dynamics (CFD) simulation using FLUENT. The analysis showed that personalized radiant system improves thermal environment near the workspace and allows all-air systems to work at higher thermostat temperature without compromising the thermal comfort, which in turn reduces its energy consumption.

DISCLAIMER:

INTRODUCTION:

Interest and growth in radiant cooling systems have increased in recent years because they have been shown to be energy efficient in comparison to all-air distribution systems (Khan et al., 2015). In radiant cooling system, temperature of the structure is reduced by supplying chilled water at lower temperature flowing through the pipes embedded in the structure.

The radiant cooling system removes the major part of the sensible load while latent load and remaining sensible load is removed by the ventilation system. The radiant cooling system can provide the comparable comfort level at higher room temperature than the conventional system at lower room temperature in context to human body reaction (Xiang et.al, 2012). Therefore, radiant cooling system, coupled with a smaller forced-air system (for ventilation, latent loads and supplemental sensible loads) can reduce a building's total energy use by operating at higher set-points.

One of the major drawbacks with the Radiant cooling system is condensation. When temperature of panel is below dew point temperature of room air, then the moisture present in the air condenses. This will result into microbial growth which decreases the air quality. Apart from this, temperature control for different systems has very different response times which lead to thermal discomfort of the asymmetrical distribution of the radiant temperature related with panels installed on ceilings and floors (Fred S. Bauman et.al., 1996).

In order to overcome this limitation many researchers designed a personalized radiant cooling system. Personalized conditioning aims to condition only a relatively small space around the user. It has been shown that such kind of systems improves an individual's comfort and reduces the energy consumption when designed and used properly (Melikov AK et.al, 2007).

It has been shown that with the use of personalized cooling system, thermal comfort can be well maintained even at the room air temperatures reaching 30 °C and at a relative humidity of 60–70% (Zhai Y et.al, 2013).

Personalized convective cooling is applied in all the reviewed studies concerning cooling, often in combination with personalized ventilation. In these studies the most common strategy for reducing the energy use is to increase the temperature set-point for the total volume conditioning while the comfort is still well maintained by the personalized cooling. The

estimated energy savings vary between 4 and 5% when the cooling set-point is increased by 2.5–6 °C (Takehito Imanari, 1999). It has to be noted that the highest energy savings of 51% are achieved by the combination of increased cooling set-point and reduced airflow rate (Schiavon S et.al, 2010).

The objectives of the present study are: 1) to localize the effect of cooling in the near vicinity of the occupant to minimize the energy used to condition the unoccupied space, 2) to understand and minimize the condensation problem on the radiant panel. In order to minimize the condensation, both, radiant and conventional cooling systems were operated in various temperature combinations maintaining thermal comfort in the vicinity of the occupant along with the minimum use of the energy. A three dimensional simulation software package, ANSYS v15.0, was used to develop the model, which was validated using test data from an existing experimental facility.

CFD simulation provides detailed spatial distributions of air velocity, air pressure, temperature and turbulence by numerically solving the governing conservation equations of fluid flows. It is a reliable tool for the evaluation of thermal environment and air distribution (Khan et al., 2015). These results can be directly or indirectly used to quantitatively analyze the indoor environment and determine system performances. A main reason of using CFD is the excellent pictorial presentations of results which allow engineers to have a better understanding on the indoor environment.

SYSTEM DESCRIPTION

Performance of radiant cooling panel with conventional air conditioning system (under transient conditions) has been evaluated in order to identify the desired temperature obtained from radiant panel and conventional air conditioning system. Numerical simulations were performed using ANSYS FLUENT v 15.0. It has the capabilities to predict the incompressible, compressible, laminar and turbulent fluid flow along with buoyancy and compressibility phenomena. FLUENT turbulence models can predict the turbulence behavior near wall via use of extended wall functions.

Description of Radiant Cubicle

The Radiant cubicle system was set up at the Centre for Energy and Environment, MNIT Jaipur (India). Figure 1 shows the schematic of the experimental setup of system. The whole experiment was only for concept testing; therefore a very simple cubicle was selected. The height of the cubicle takes as 1.4 m considering the average distance of the head from the ground of a 1.8 m person.

Figure 1 Experimental setup of radiant cubicle

The chilled water is circulated through copper tubes and the heat transfer from the working fluid is setup by radiation and conduction. The tubes are mounted on the metal sheet by riveting. However the contact area available for heat transfer from the fluid to the sheet is limited due to the line contact. Thus, a thin Aluminum sheet (<0.5 mm) is used to wrap the copper tubes and extended at the ends in the shape of a fin to provide efficient heat transfer. The room in which radiant cubicle is situated is a single story room (13 m^2) with concrete exterior walls and floor and having a conventional air-conditioning system in order to handle the latent load.

THE CFD MODEL

Model Setup

The physical geometry of radiant cubicle system with surrounding room were modeled (Figure 2) keeping all the geometrical parameters same as in the existing experimental set up using ANSYS's workbench platform i.e. ANSYS's DESIGN MODELER.
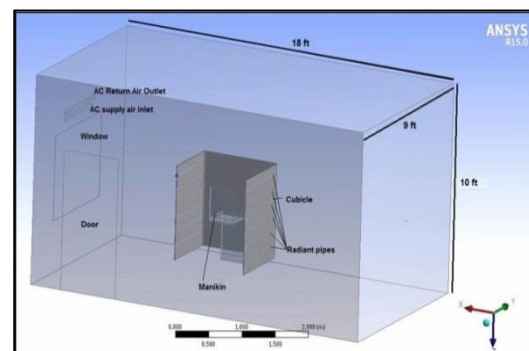

Figure 2 Model of cubicle in room with background AC

Meshing Details

Developed physical model of building was meshed using 3D hexahedral meshing comprised of total 1987832 cells (elements) at a growth rate of 1.2, in ANSYS's workbench MESHING total. The meshed drawing is shown in figure 3.
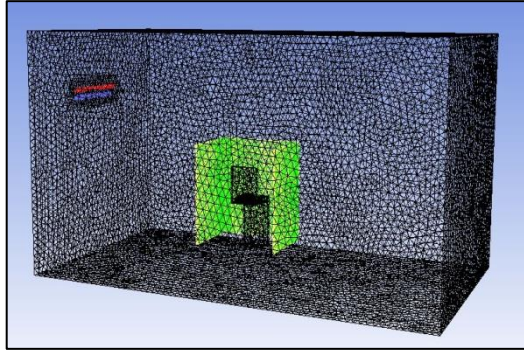
Figure 3 Meshing geometry of room

ANSYS FLUENT v15.0 was used in the study that used the finite volume method to convert the governing equations in to numerically solvable algebraic equations. The numerical investigations were based on the following assumptions:

i.   Thermo-physical properties of aluminum, concrete, human and air remain constant during operation.

ii. Thermal contact between chilled water pipe and its surrounding is perfect.

iii. Initially, room air temperature, chilled water pipe temperature and panel temperature is considered equal and undisturbed.

Modelling of thermal manikin

The manikin was placed in the middle of radiant cubicle with the floor dimensions 5.5 m×2.8 m and celling height of 3 m. The boundary conditions at the thermal manikin surface were considered as a constant. The uniform heat flux from the surface was set to 55 W/m^2 (Petr Zelensky, 2013), which yielded the total sensible heat output of 90 W.

Initial and boundary conditions

The far-field distances were carefully selected to ensure that the far-field boundary temperature would not have any effect on simulations. Therefore, far-field boundaries in the room environment were treated as adiabatic surfaces. Cubicle wall and surrounding air were 'Coupled' in order to enable heat transfer and their initial temperatures were set according to the experimentally measured values.

1) Indoor parameters: Dry bulb temperature at 27 °C and relative humidity of 60%.

2) Internal Walls and floors: Heat transfer is assumed as constant heat flux boundary condition and heat transfer density is 17 W/m^2 (Petr Zelensky, 2013).

3) Outside Window: Window is made of glass with its adequate properties and heat transfer coefficient.

4) Air inlet: Air conditioning air supply inlet is simplified as rectangular air supply outlet which is set below the air conditioning panel. 'Velocity inlet' boundary condition was specified for the inlet air velocity of 3 m/s and supply air temperature kept constant for 18 °C.

5) Chilled water inlet: For simplicity of model, the temperature of radiant pipe kept constant for a case throughout the simulation.

6) Air outlet: 'pressure outlet' condition specified for return air outlet.

SIMULATION AND NUMERICAL SETUP

Based on essential principle, CFD requires a discretization of a physical field continuous in time and space originally. This includes temperature fields, pressure field and velocity field, which are represented by a collection of variable and different models. The setting of CFD simulations are shown in Table 1. The model includes specific sub-models which are described hereafter.

Turbulence sub-model

As the air flow was assumed to be turbulent, a k-ε model was considered for the turbulence model. The k-ε standard model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The k-ε model is commonly used in most CFD programs for building (AWBI H.B., 1991). This model is useful as the requirements of computer hardware are not so demanding and applicable for a wide range of flows. The k-ε model would not give results much different from those predicted by more complicated models with second order correction; nor with combination of models (W. Szablewski, 1972).

Condensation sub-model

A two-phase model has been formulated in the Mixture model framework in FLUENT. Moist air has been considered as the primary phase, with the water as the secondary phase. Interphase mass transfer has been tracked by using the thermal phase-change model. The model for the surface condensation of the water vapor is based on the following assumptions: the humid air is an ideal gas, being a perfect mixture of 2 perfect gases (dry air and water vapor); there is no chemical reaction between these two gases; the binary mixture is an incompressible fluid, the heat and mass transfer interactions are negligible; the quantity of condensed vapor is low (as it usually happens in buildings); the amount of condensate is removed from the computational domain.

Table 1: Solver setting

Initial condition	Value
Velocity, temperature, kinetic energy, dissipation rate and radiation	Deduced from steady state simulations using the boundary conditions values
Water vapor mass fraction	Calculated from recorded relative humidity
Boundary condition	Values
Manikin properties	Material, constant heat flux
Material used	Concrete, human body, aluminum, copper
Inside radiant pipes	Constant temperature
Solution methods	PISO pressure velocity coupling
Model used	Turbulent model, multiphase model, radiation model, species transport model

CONVERGENCE AND GRID INDEPENDENCE TEST

The convergence criteria were achieved for continuity, x-velocity, y-velocity, z-velocity, $k - \varepsilon$ ($1e^{-3}$) and the energy equation ($1e^{-6}$) at the each time step. The grid independent size was determined by increasing the number of meshes until the criterion $\frac{\dot{p} - \dot{p}+1}{\dot{p}} < 10^{-3}$ was satisfied. Here $\dot{p}$ represents the calculated temperature using current mesh size and $\dot{p} + 1$ correspond to temperature using the next mesh size. For the grid independent, the room air temperature was evaluated.

MODEL VALIDATIONS

Performance of cubicle has been evaluated for several days for different configuration of supply air set point temperature of air-conditioning system and chilled water temperature. The temperature in the vicinity of occupant and room interior air temperature were measured at every 1 h interval. The air velocity of supply air was also measured.

For the validation of model, one operational day was considered to validate the correctness and reliability of the CFD model. The initial room air temperature was measured and collected using a data acquisition system for the entire duration of operation. The AC inlet air flow velocity was kept constant at 4 m/s during the whole operation time and temperature of inlet air was varied at every 30 min interval according to measured temperature through PROFILE function capturing the transient behavior of air. The purpose was to analyze the temperature in the vicinity of occupant and condensation effect study.

Figure 4 and figure 5 shows the variation in measured and simulated interior air temperature and air temperature in vicinity of occupant. It can be seen that a good agreement exists between measured and simulated temperatures. The maximum temperature difference observed was 0.6 ^{o}C. This agreement establishes the validity of proposed model used for the simulation and was considered appropriate for further analysis.
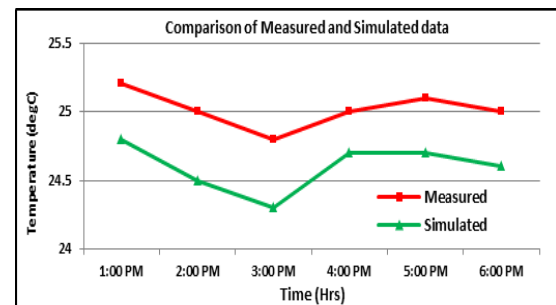

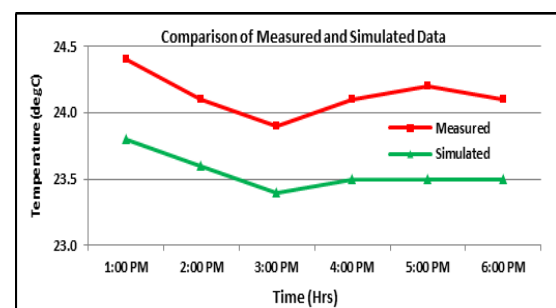*Figure 4: Validation of average air temperature for a day*

Figure 5 Validation of air temperature in the vicinity of occupant

METHODOLOGY FOR PARAMETRIC ANALYSIS

Based on the validated CFD model, three different operation modes have been investigated in order to increase the thermal performance and minimize condensation problem of radiant cubicle. The different radiant chilled water combination will be

useful to study the condensation problem. Figure 6 shows the flow chart of parameters of radiant system which is studied in present work to parametric analysis. The effect of variation in parameters of system is analysis on the basis of thermal performance of system. The variation in temperatures of AC supply air 24 °C, 26 °C and 28 °C, at different chilled water temperature 14 °C, 15 °C and 16 °C is considered for 1 h continuous operation in room.
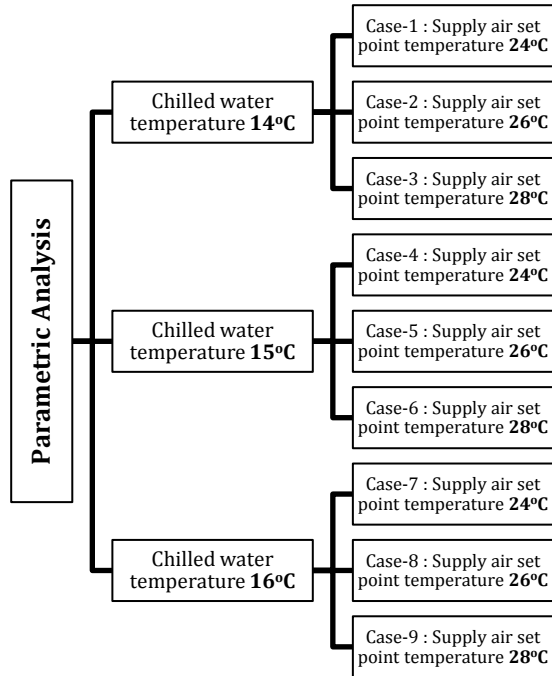
```
                              ┌──────────────────────────────┐
                         ┌────┤ Case-1 : Supply air set      │
                         │    │ point temperature 24°C       │
         ┌─────────────┐ │    ├──────────────────────────────┤
         │ Chilled water├─┼────┤ Case-2 : Supply air set      │
         │ temperature  │ │    │ point temperature 26°C       │
         │ 14°C         │ │    ├──────────────────────────────┤
         └─────────────┘ └────┤ Case-3 : Supply air set      │
                              │ point temperature 28°C       │
  Parametric Analysis         ├──────────────────────────────┤
         ┌─────────────┐ ┌────┤ Case-4 : Supply air set      │
         │ Chilled water│ │    │ point temperature 24°C       │
         │ temperature  ├─┼────┤ Case-5 : Supply air set      │
         │ 15°C         │ │    │ point temperature 26°C       │
         └─────────────┘ └────┤ Case-6 : Supply air set      │
                              │ point temperature 28°C       │
         ┌─────────────┐ ┌────┤ Case-7 : Supply air set      │
         │ Chilled water│ │    │ point temperature 24°C       │
         │ temperature  ├─┼────┤ Case-8 : Supply air set      │
         │ 16°C         │ │    │ point temperature 26°C       │
         └─────────────┘ └────┤ Case-9 : Supply air set      │
                              │ point temperature 28°C       │
```

Figure 6: Methodology flowchart

NUMERICAL RESULTS AND DISCUSSIONS

The CFD simulations were carried out with the available package, ANSYS FLUENT v15.0. Preliminary simulations were undertaken to get grid independence results. All simulations have been continued for 3,600 seconds with a refined mesh in the region across the cubicle with a 1e-2 time step size. The convergence criteria were fixed to 10^{-3} for the continuity, turbulent kinetic energy and dissipation rate, whereas they were equal to 10^{-6} for the energy, radiation and water vapor mass fraction. Convergence was reached after about 1,200 iterations at each time step initially.

Analysis of indoor temperature

It is important for air conditioning system to ensure the rationality of indoor air distribution because it is closely related to the effect of adjusting indoor temperature, human body comfort.

In this study, the temperature distributions are presented based on only the CFD simulation using three different chilled water temperature 14 °C, 15 °C and 16 °C for three different supply air inlet

temperatures 24 °C, 26 °C and 28 °C for constant inlet velocity. Figure 7 shows the variation of temperature for different cases. It can be seen that second case in which air conditioner set point temperature is 26 °C can maintain the indoor air temperature in the comfort range. And also, the choice of chilled water temperature depends upon the condensation which eventually depends upon the latent load.
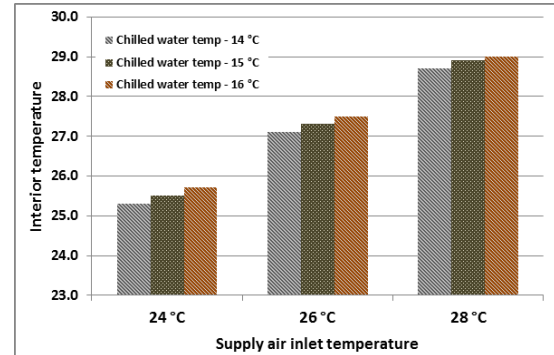

Figure 7 Analysis of indoor air temperature

Figure 8 show the temperature gradient of indoor temperature along a cross section for a particular time and it has been seen that there is a difference between temperatures in the vicinity of occupant as compared to room.
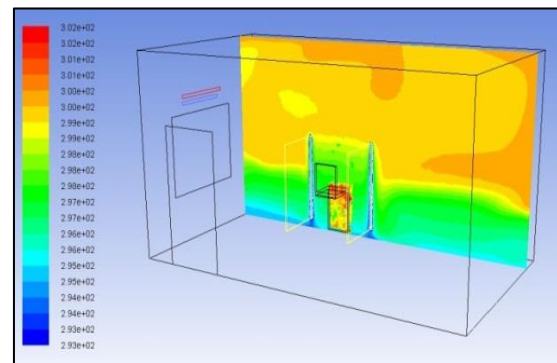

Figure 8: Temperature gradient along a cross section

Analysis of air movement

The output result indicates that the velocity of air due to convection varies uniformly. Following is the snapshot of results for velocity of air across a cross-section of room. The velocity of air remains undisturbed for all combinations of temperatures. Figure 9 shows the velocity vectors for a particular case for a particular time.
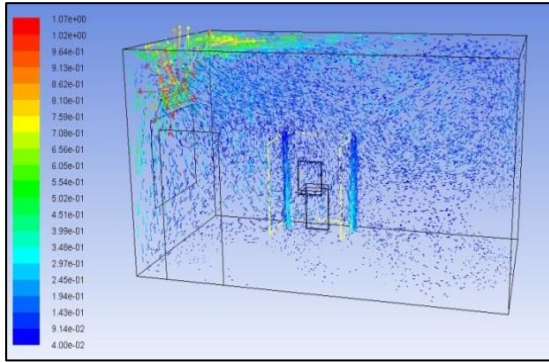
Figure 9: Temperature gradient along a cross section

Analysis of temperature in the vicinity of occupant

Figure 10 indicates (red horizontal line) that the same comfort temperature range near the occupant can be achieved with higher set point temperature of air conditioning system which saves the energy. These radiant panels in the vicinity to control the microclimate around the occupant help to increase energy saving without disturbing thermal comfort of occupants.
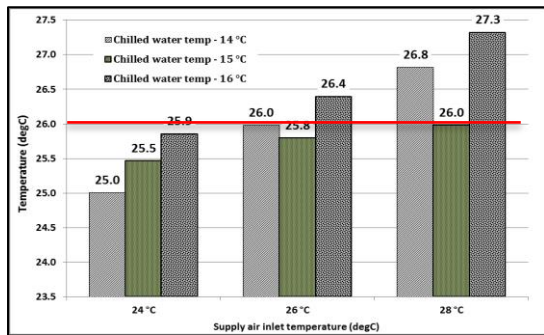

Figure 10: Temperature gradient along a cross section

Following figure 11 is the representation of temperature contour of the room at working height of occupant. It can be shown that the temperature is lower in the radiant cubicle compared to the room.
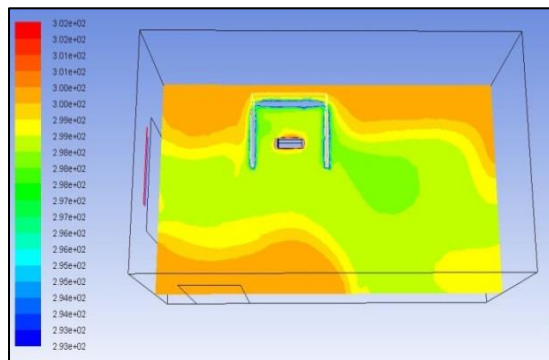

Figure 11: Temperature gradient along a cross section

From figure 12 it has been shown that there is some difference of around 0.7 °C between the interior air temperature and temperature in the vicinity of occupant in maximum cases. This difference leads to work on same operative temperature with increasing set point temperature of background air-conditioner. This means, higher set point temperature of air conditioner can be achieved the same level of comfort using personalized radiant cooling.
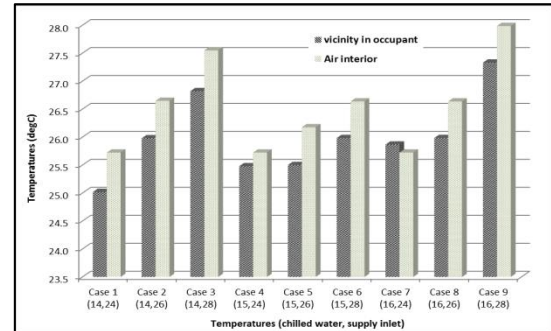

Figure 12: Temperature representation of interior air and vicinity of occupant for all cases

Analysis of condensation on radiant cubicle

Based on the CFD analysis, interfacial area for condensation on radiant cubicle has been obtained. The moisture condenses at gas–liquid interface i.e. the pipes of radiant cubicle. As the phenomenon of condensation is transient in nature, it is important to track the interface and hence the available interfacial area for heat transfers. The results show that condensation is maximum when chilled water temperatures are minimum. Condensation rate decreases with increase in chilled water temperature. From figure 12, it has been found that with increasing in both temperatures of chilled water and background air-conditioning, condensation decreases without affecting thermal comfort of occupant.
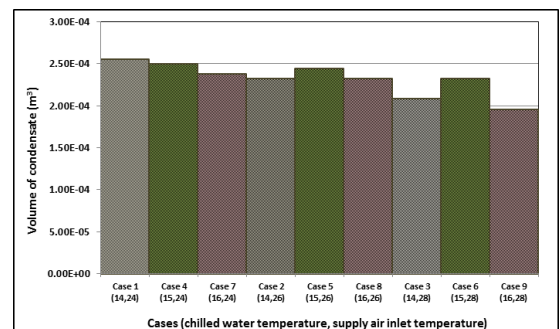

Figure 12: Condensate analysis for different cases

CONCLUSION

An individual approach to the building occupants makes it possible to satisfy different needs of different persons and thus improves comfort and subsequently performance. The study includes an office cubicle made with the radiant panels which itself is made up of Aluminum and the looping inside in it is of Copper pipes. Using radiant panels

in the near vicinity to control the microclimate around the occupant, the set point temperature of the background air-conditioner can be increase for the same operative temperature which leads to energy saving.

Different combinations were simulated with help of CFD tool FLUENT by varying chilled water temperature and supply air temperature of background air-conditioner. The results show that the air temperatures in the room are quite uniform under three air supply modes and the average temperature difference between temperatures in the vicinity of occupant and interior is around 0.7 °C.

FUTURE SCOPE

Since the whole experiment is for concept testing, therefore a very simple cubicle design was selected as discussed earlier. This model needs improvement to increase energy saving opportunities which can be done by study the performance on cubicle integrated with Chillers and DX-coils.

ACKNOWLEDGEMENT

REFERENCES:

AWBI H.B., Ventilation of Buildings, E&FN Spon, 1991.

Melikov AK, Knudsen GL. Human response to an individually controlled microenvironment. HVAC&R Res 2007;13:645–60.

Petr Zelensky, Martin Burtak, Jan L. M. Hensen. Simplified representation of indoor heat sources in CFD simulation. 13th Conference of International Building Performance Simulation Association, Chambéry, France, August 26-28.

Schiavon S, Melikov AK, Sekhar C. Energy analysis of the personalized ventilation system in hot and humid climates. Energy Build 2010;42:699–707

Takehito Imanari, Toshiaki Omori, Kazuaki Bogaki, Thermal comfort and energy consumption of the radiant ceiling panel system. Comparison with the conventional all-air system. Energy and Buildings 30 1999 167–175.

Task/Ambient Conditioning Systems: Engineering and Application Guidelines by Fred S. Bauman and Edward A. Arens, Centre for Environmental Design Research, University of California, Berkeley, CA, October 1996.

W. Szablewski, in: B.E. Launder, D.B. Spalding (Eds.), Mathematical Models of Turbulence, vol. 169, S.M. Abb, London/New York, 1972.

Xiang-Long Liu, Guang-Cai Gong, Heng-Sheng Cheng and Li-Xing Ding. Airflow and Heat Transfer in the Slot-Vented Room with Radiant Floor Heating Unit. Journal of Applied Mathematics, Volume 2012, Article ID 287271.

Y. Khan, V. Khare, J. Mathur, M. Bhandari, Performance evaluation of radiant cooling system integrated with air system under different operational strategies, Energy and Buildings 97 (2015), 118–128.

Zhai Y, Zhang H, Zhang Y, Pasut W, Arens E, Meng Q. Comfort under personally controlled air movement in warm and humid environments. Build Environ 2013;65:109–17.