STUDY OF CONVECTIVE HEAT TRANSFER IN A RADIATIVELY COOLED BUILDING USING COMPUTATIONAL FLUID DYNAMICS

Vinay Ghatti
Sayyed Mohammed Autif
School of Architecture
Arizona State University
Tempe, AZ. 85287-1605
ghatti@hotmail.com
smautif@hotmail.com

ABSTRACT

All material objects emit heat in the form of infrared radiation. The intensity of this radiation depends on the temperature and emissivity of the object. Although the majority of heat transfer in any radiative system is through radiation, convection too forms an integral part of heat transfer due to differences in surface temperatures. The process of convection not only affects heat transfer, but also helps maintain comfort conditions. This paper deals with the convective component of radiative cooling, which is the basic principle in some of the passive solar strategies like 'roof pond', 'night-sky radiation', etc. and is increasingly being considered as the primary space-cooling device in buildings. According to a preliminary estimate, the contribution of convection in the space cooling process could be up to 40%.

This paper explores the convective heat transfer in a radiatively cooled high-mass residence at Arizona, as part of an ongoing research project funded by the ASHRAE. This is done by using Computational Fluid Dynamics to study the convective effects in an enclosure with radiantly cooled ceiling system on a typical summer day. The study is conducted for a typical summer day using actual data recorded at site. The research explores issues like- the effect on air movements, air speeds, stratification, and comfort conditions due to variations in ceiling, wall and floor temperatures. The research addresses some of the critical aspects of passive solar strategies applicable in various situations.

1. INTRODUCTION TO RADIANT COOLING

Most prevalent air-conditioning systems are forced air (also known as convective systems), where the room air is recirculated, chilled and blown back into the space. With higher space loads, more air needs to be blown in, resulting in uncomfortable cold drafts. The recirculation of the air can potentially result in indoor air quality problems, since harmful organisms/ pollutants can be spread easily.

Radiant cooling technology is generally more energy-efficient than the forced air systems, and it avoids the problems mentioned previously, resulting in greater human comfort. The system separates the cooling and ventilation tasks of a building conditioning system. Some other advantages of a radiant cooling system are:

- Simplification of wall, floor and structural systems, since space-conditioning equipment is not needed at the outside walls.
- Flexibility for later space partitioning.
- Simplified maintenance and operation, since all mechanical equipment can be centrally located.
- Noise free space cooling.
- Reduction in peak loads due to thermal storage.

The biggest disadvantage with the radiant cooling system is condensation with in the space. In humid climates, this is a major restriction for using this system.

2. COMPONENTS OF RADIATIVE COOLING

In a radiant cooling system, heat exchange between the radiator and the rest of the room happens mainly via radiation and convection. The warmer objects having a view of the cold radiator radiate their heat to it. The radiant heat transfer is governed by the Stefan-Boltzmann equation

$$q_r = 0.15x10^{-8} [(t_p+460)^4 - (AUST+460)^4]$$

where,
q_r = radiant cooling, Btu/h·ft2
t_p = effective panel surface temperature, °F
AUST = area-weighted average temperature of uncontrolled surfaces in room, °F

Convection takes place when the air passing the radiant panels cools down and becomes heavier. This creates natural, high volume, low velocity air currents. According to some research, 40-50% of the radiant cooling effect is typically due to convection (2). Convection in a panel system is a function of the panel surface temperature and the temperature of the airstream layer directly below the panel. (3).

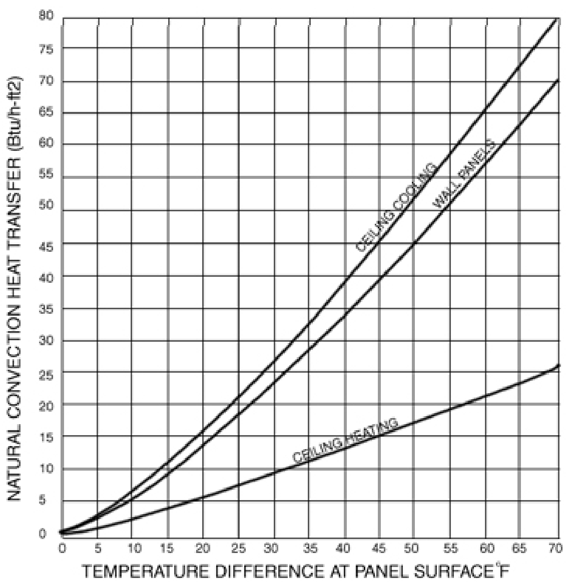

Fig. 1 Natural Convection Heat Transfer at Floor, Ceiling and Wall Panel Surfaces. (Source ASHRAE, S&E 1996)

The following equations are used to calculate heat transfer from radiant cooling panels by natural convection (ASHRAE 1996):

$q_c = 0.31(t_p - t_a)^{0.31}(t_p - t_a)$ –for cooled ceiling panels
$q_c = 0.26(t_p - t_a)^{0.32}(t_p - t_a)$ –for cooled wall panels

where, t_a= indoor air design temperature (under normal conditions).

There is no confirmed data for floor cooling panels, but the following equation can be used for approximate calculations:

$q_c = 0.13(t_p - t_a)^{0.25}(t_p - t_a)$

3. THE CFD SIMULATION TOOL

The convective flow in a radiantly cooled room can be analyzed using fluid dynamics. It can be done by using physical experiments, or through the use of Computational Fluid Dynamics (CFD). Physical experiments tend to be very expensive and tedious. CFD is a relatively easier alternative, especially since the computing power of desktop computers has improved drastically over the last decade or so. CFD is commonly accepted as referring to the broad topic encompassing the numerical solution, by computational methods, of the governing equations which describe fluid flow, the set of the Navier-Stokes equations, continuity and any additional conservation equations (4). CFD not only predicts fluid flow behavior, but also the transfer of heat, mass, phase change, chemical reaction, mechanical movement, and stress or deformation of related solid structures. Most CFD tools provide sophisticated visualization options, which makes it easier to analyze several complicated phenomena.

The CFD tools are being increasingly used in the industries concerned with fluid flow. Some programs have been developed for the HVAC industry as well. For the purpose of this study, the AirPak CFD tool from Fluent Inc. was used.

The AirPak program is a customized version of the Fluent program. It lets the user accurately model airflow, heat transfer, contaminant transport, and thermal comfort in ventilation systems as well as external building flows.

The program uses a new zero-equation indoor turbulence model that addresses the need for a simple but reliable turbulence model for room ventilation (5). The program is capable of importing CAD data in DXF and IGES formats. The program interface is user friendly and allows quick set up of the model and its specifications.

4. THE CFD MODEL

4.1 A Description of the Radiant House

Located in Carefree, Arizona, the high-mass residence built with adobe, has insulation on the exterior and radiant panels in both the ceiling and the floor supplied by a hydronic source, with a ground source hear pump. The house is a single story slab on grade of approximately 2500 sq. ft. (250 $m^{2)}$, with 14-inch (30cm) adobe exterior walls and sloping roofs without attic space. The ceiling panels are used for summer cooling and floor panels are used for winter heating.

Fig. 2 The plan of the residence marked with the simulated zone.

The control strategy has been designed to use the mass in walls and floor for thermal storage so as to keep the spaces within the comfort envelope while using the minimum amount of on-peak energy. The residence is divided into 3 zones, east, center and west.

4.2 The Envelope

Only the west zone, consisting a master bedroom along with a toilet block is used for the simulation purposes. Measuring 6m x 9m in plan, the room has a pitched roof rising from 2.4 m to 3.5m at the ridge. The entire room is enclosed within a rectangular block of 6m x 9m x 4m. The room has 2, 1 and 2 windows on the west, north and south side respectively. The east part of the room has interior walls with a toilet block.

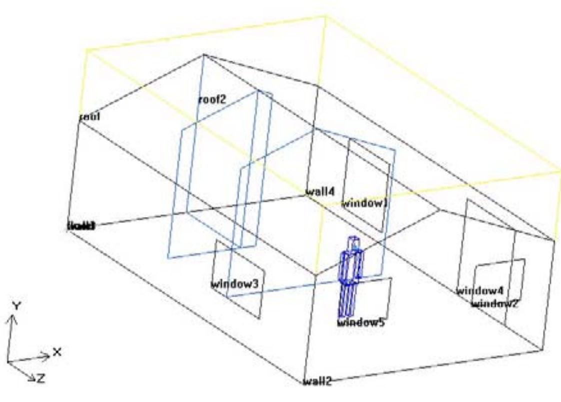

Fig. 3 Simulated model: Master bedroom

The exterior walls are modeled as 0.39m thick with-
Solid material: 14-inch adobe
Exterior surface: Asbestos board
Interior surface: Paint-white-acrylic

Due to the limitations of the simulation tool, the exterior insulation was replaced with asbestos board and the interior adobe-plaster was replaced with white paint. The double paned, green glazing is replaced with heat resistant plastic surface with following properties-
Roughness: 0.0m
Emissivity: 0.9

4.3 The Radiant Floor

The slab on grade floor contains 3/8" dia (1cm.) rubber tubing at a spacing of 9-inch (22.5cm). Finished with flagstone, the flooring has 2-inch (5cm) insulation around perimeter. The concrete slab on grade is modeled as-
solid material: concrete
External surface: Asbestos fiber
Interior surface: Ceramic

Due to the limitations of the tool, the exterior dirt was replaced with asbestos fiber and the interior flagstone was replaced with ceramic.

4.4 The Radiant Ceiling

The ceiling contains capillary tube matting made of polypropylene. The 1/12-inch (2mm) ID capillary tubes are spaced at 4/5 inch (20 mm). The low-mass roof is modeled as 0.21m thick with-
Solid material: polyurethane (R21)
Exterior surface: steel-Polished-surface
Interior surface: paint-white-acrylic

Due to the limitations of the simulation tool, the air space separated by a radiant barrier could not be incorporated in the roof layer. The interior cement plaster was replaced with acrylic paint. The roof is assigned a radiant temperature of 16.5 ^{o}C to mimic the embedded radiant panels for cooling.

4.5 Interior Partition Walls

The interior partitions are modeled as light weight adiabatic walls. Due to the limitations of the simulation tool, the interior partitions could not be modeled as wood frame partitions and adobe partitions.

4.6 Occupant

A person was located within the room to study the comfort conditions. The properties of the person are-
Posture: standing
Clo. Value: 0.5
Metabolic rate: 1.0

5. SIMULATION

The following are the various parameters considered for the simulation-

Time variation: Steady state
Variables solved: Flow, Temperature, Radiation, Comfort
Flow regime: Turbulent
Gravity Factor: -9.8m/s^2
Ambient Temperature: 109^{o}F (43^{o}C)
Initial velocity: x: 0 m/s, y: 0.00098m/s & z: 0 m/s
Initial Temperature: 77^{o}F (25^{o}C)
No. of iterations: 100

Following are the properties of the mesh-
Grid Type: Hexa
Max x-size, y-size, z-size: 0.2
Min. elements in fluid gap: 3
Min. elements in solid edge: 2
Max size ratio: 2
Max O-grid ht.: 0
Rayleigh No. 3.6427e+011
Prandtl No. 0.74311859504132

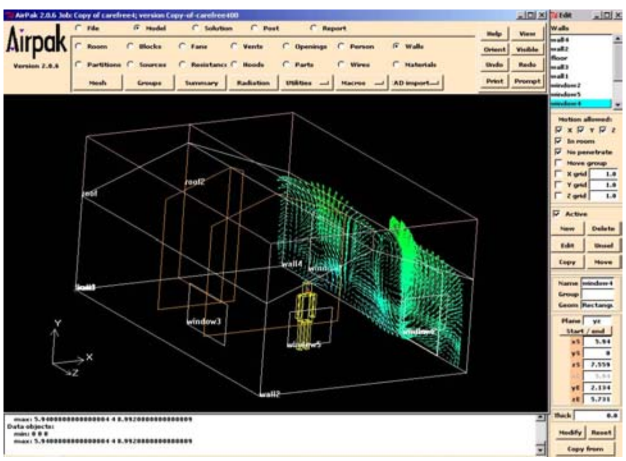

Fig.4 A snap shot of user-interface

6. OUTPUT

6.1 Analysis of Indoor Temperature

The output results indicate that the temperature varies around 77^{o}F (25^{o}C) and there is no effect of stratification due to continuous convection within the room. The temperature is highest along the faces of the windows, which contribute to maximum heat gain within the enclosure.

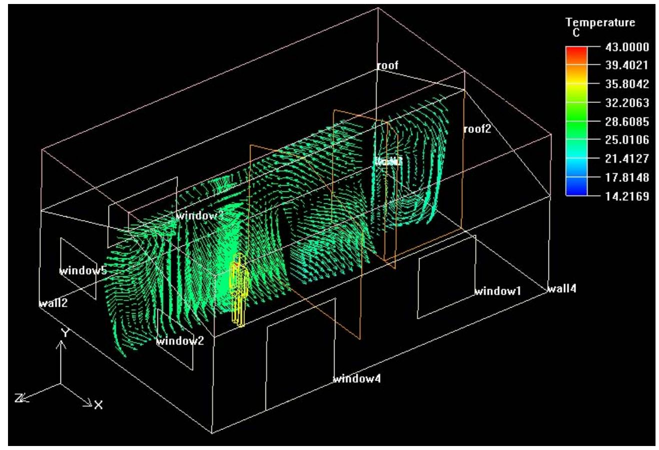

Fig.5 Temperature gradient along a cross section.

6.2 Analysis of Air Movement

The output results indicate that the velocity of air due to convection varies from o m/s to 0.6 m/s. Following are the various snapshots of the results for velocity of air across different cross sections of the room-

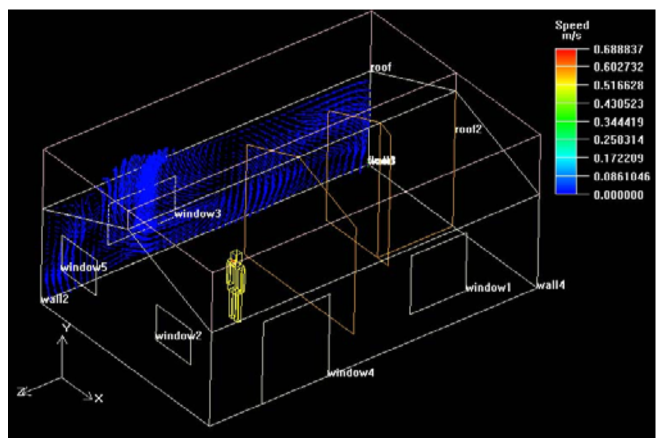

Fig.6 Velocity gradient along the room

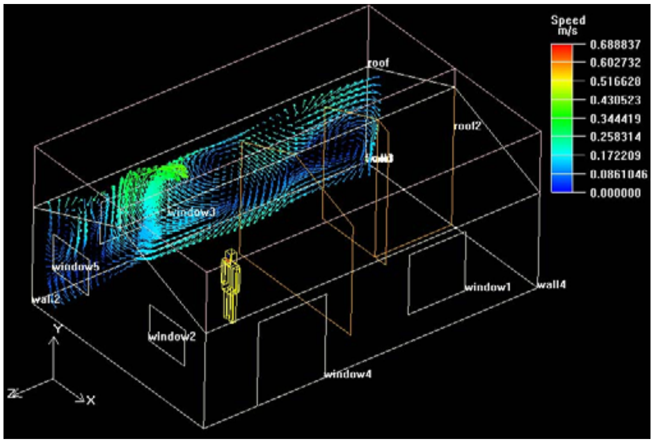

Fig.7 Velocity gradient along the room

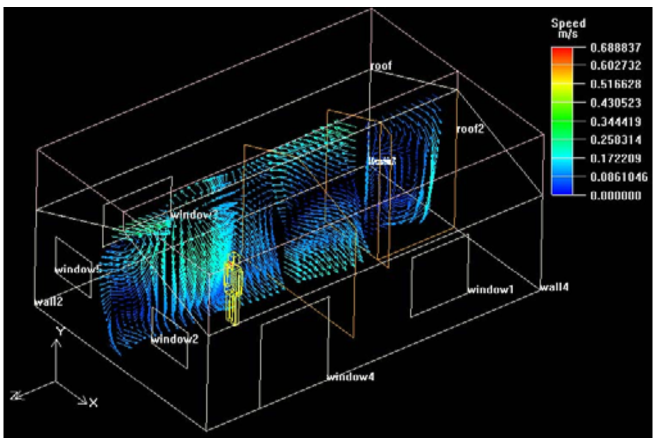

Fig.8 Velocity gradient along the room

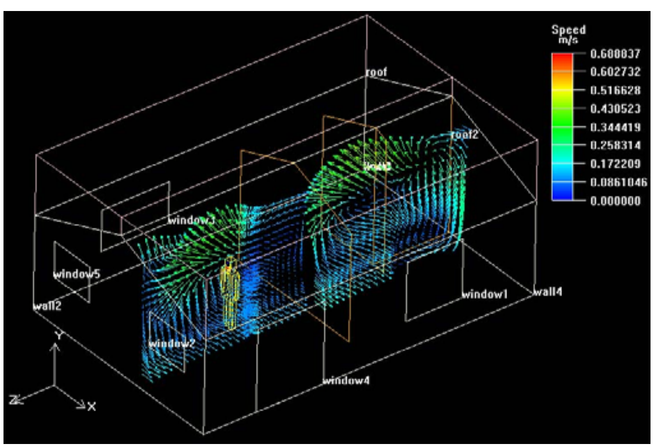

Fig.9 Velocity gradient along the room

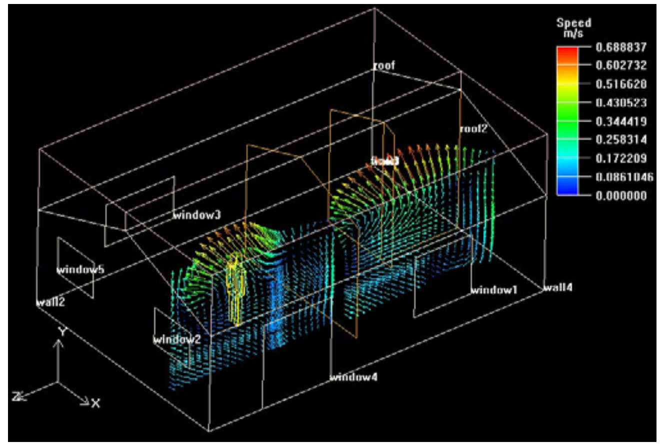

Fig.10 Velocity gradient along the room

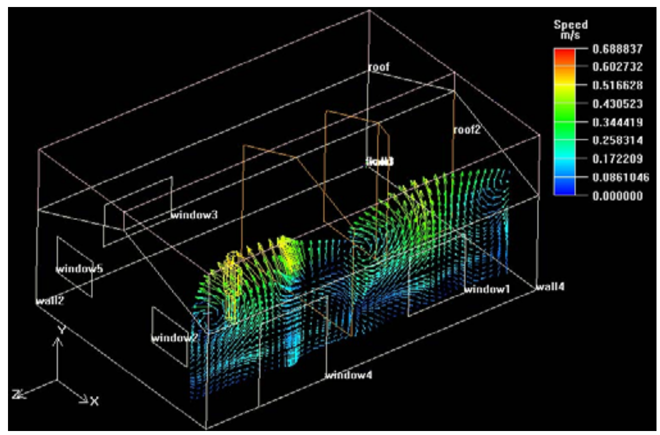

Fig.11 Velocity gradient along the room

7. CONCLUSION

The following conclusions are strictly based on the results of the simulation, hence are bound by the limitations and capabilities of the simulation tool-

As per the simulation results, a cool ceiling generates considerable convection to achieve indoor comfort. A cooled ceiling absorbs heat form the surrounding air, thus making it heavier and dropping down, which is replaced by the rising hot air. The convection within the room increases if the windows are assigned with a wind flow. The temperature within the room varies around 77^{o}F (25^{o}C). The corresponding air velocities, which are generated, vary from 0m/s to 0.68 m/s. The air velocity is lowest along the surfaces of the walls and increases gradually towards the center of the room. The air velocity is highest around the person, as the heat generated by the person heats up the air and pushes it upwards. The air velocity increases as the difference between the ceiling temperature and the ambient temperature increases, and also when the windows are opened and a natural airflow is assumed through the openings.

8. REFERENCES

(1) Sprecher,F., “An introduction to radiant cooling technology,” May 1999

(2) Conroy, C.L.and Mumma, S. A., Ceiling Radiant Cooling Panels as a Viable Distributed Parallel Sensible Cooling Technology Integrated with Dedicated Outdoor Air Systems

(3) ASHRAE. 1996. “1996 ASHRAE Handbook—HVAC systems and equipment”, Atlanta: American Society of Heating, Refrigeration and Air-conditioning Engineers

(4) http://www.cranfield.ac.uk/sme/cfd/ "What is CFD?"

(5) Fluent Inc. 2001 “Getting Started with Airpak2”