International Journal of Heat and Mass Transfer 87 (2015) 71–78

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Experimental and numerical study of a small-scale and low-velocity indoor diffuser coupled with radiant floor cooling

A. Fernández-Gutiérrez, I. González-Prieto, L. Parras, J.M. Cejudo-López, C. del Pino *

Universidad of Málaga, Andalucía Tech, Escuela Técnica Superior de Ingeniería Industrial, 29071 Málaga, Spain

ARTICLE INFO

Article history:
Received 9 August 2014
Received in revised form 19 March 2015
Accepted 19 March 2015
Available online 11 April 2015

Keywords:
Displacement ventilation
CFD
Thermal modeling
Radiant floor

ABSTRACT

We present results of one small scale displacement diffuser coupled with radiant floor cooling. Three control parameters are changed, both numerical and experimentally: flow rate, temperature difference between the undisturbed atmosphere and the supply air, and between the floor and the supply air. Flow visualizations and measurements of the velocity field are carried out using Particle Image Velocimetry for a laminar, axisymmetric, and steady state flow. Numerical simulations show an excellent agreement with experimental data. Special attention is paid to the vicinity of the floor. The main finding in this experimental setup is the appearance of a structure similar to a lee wave, typical in atmospheric flows. This standing wave, named thermal jump in this work, appears while forming a recirculation bubble in a stratified atmosphere with the eruption of the cool air flow that has greater temperature than the ground. To our knowledge, this is the first reported experimental evidence that produces a thermal jump under steady state conditions in air without the breakdown of the plume. Finally, we characterized the heat transfer as function of non-dimensional parameters, showing a parabolic correlation in terms of the radial distance.

1. Introduction

Air conditioning in large spaces with floor heating or cooling systems together with displacement ventilation is a recurrent research topic [1]. Displacement ventilation systems improve thermal conditioning supplying air at low speed, its basic principle being the variation of air density, enhancing a vertical temperature gradient from the floor to the ceiling [2–4]. Its effectiveness increases within the occupied zone, from the ankles to the head. If the supply air has also low moisture levels, the dew point decreases and, consequently, it increases the cooling capacity of the floor. From an engineering and practical design points of view, it is appropriate the use of radiant floor coupled with displacement ventilation for cooling in buildings, specially in warm areas [5,6]. In the last two decades, there are research works dealing with systems in large areas using displacement ventilation with radiant floors coupled for heating or cooling, e.g. multistory buildings, hospitals, big stores, airports and train stations (see [7] or [8] and the references therein). Recently, there are works focused on the measurements of temperature and velocity fields [9,10]. In addition, features such as the air temperature distribution, the temperature of the floor and the air velocity determine the comfort sensation in these systems. These thermal parameters are important to control the cooling system [11–13], to develop complex building energy simulation software [14,15] or to revise the compliance with standards [16]. However, one of the main disadvantages of cooling floors is the risk of condensation [17] that is also present in chilled ceilings [18–20].

Turbulence appears as loads (person or equipments) interact with the treated air flow [2]. In the case of cooling, experimental measurements show among other recent research works, that the indoor air flow becomes fully turbulent in a short distance downstream from the diffuser [21–23]. Though appropriate turbulence models are used in the State of Art and much effort has been put into the improvement of meshing algorithm in the last three decades [24–28], accurate numerical simulations are difficult to obtain due to turbulence modeling. Experimental data are always required to validate numerical results. Besides, turbulent air flow properties and physical parameters must be estimated together with the presence of complex geometries [29–32]. Many works have carried out simulations to simplify turbulent models or boundary conditions [33–38]. Nevertheless, turbulence [39] or plume generation [40,41] are out of the scope of this study, therefore in this work we present a laminar regime for cooling purposes using a radiant floor in a simple experimental arrangement. In fact, the laminar case corresponds to the exact solution of Navier–Stokes equations

* Corresponding author. Tel.: +34 951952429; fax: +34 951952605.
E-mail address: cpino@uma.es (C. del Pino).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.03.068
0017-9310/

Nomenclature

α	thermal diffusion (m^2/s)	Nu_I	Nusselt number
β	thermal expansion coefficient (K^{-1})	Pr	Prandtl number
ν	kinematic viscosity (m^2/s)	Q	flow rate (m^3/s)
ρ	air density (kg/m^3)	q_I	integrated heat flux (W)
θ	non-dimensional parameter	r^*	non-dimensional radial distance
θ_T	non-dimensional temperature	R_o	outer radius of the diffuser (m)
c_p	specific heat at constant pressure (J/(kg K))	Re	Reynolds number
g	gravity acceleration (m/s^2)	T_a	ambient temperature (K)
Gr	Grashof number	T_f	floor temperature (K)
h	annular slot length of the diffuser (m). Characteristic length	T_{in}	supply air temperature (K)
		V^*	non-dimensional velocity
H_o	height of the diffuser (m)	V_c	characteristic velocity (m/s)
k	thermal conductivity (W/(m K))	z^*	non-dimensional axial distance

with no modeling. Hence, a (simple) laminar-flow model can provide a first approximation of the air flow behavior. This fact eases a comprehensive knowledge of fluid dynamics to shed some new light on this fundamental problem to provide guidance for optimizing ventilation system designs [42].

On the other hand, there are lee waves that appears in atmospherical flows when the topography of the ground is changed [43]. Though our geometry has a flat floor, we introduce this type of waves due to the flow patterns observed in our experimental setup with supply air temperature greater than the floor one. Mostly of these experimental evidences in the State of Art are studied using salt stratified water instead of fresh water to analyze temperature stratification [1,44,45]. Other theoretical studies focus on the main question of how the attached cool plane jet velocity field behaves near the vicinity of the plate floor at high Reynolds numbers [46]. When the floor temperature is lower than the temperature of the supply air, the boundary layer produces a buoyancy effect that generates an adverse pressure gradient in the vertical direction that may separate the attached cool jet. This phenomenon occurs when the Froude number is sufficiently small, leading to a separation bubble that covers a fraction of the plate depending on the Prandtl number. Without thermal boundary breakdown, the resulting bubble of the laminar air flow becomes a steady thermal jump, having a similar shape to that given in a hydraulic jump. These air flow patterns have been also reported using a boundary layer approximation [47]. In this work, we present novel results of the separation bubble that covers a fraction of the floor by using laminar air flow instead of a stratified salt water tank.

A numerical approach based on computational fluid dynamics (CFD-Fluent code was used) and experimental techniques were also applied to obtain qualitative and quantitative measurements. The present work is organized as follows. Section 2 begins with a description of the experimental arrangement together with the numerical method. Subsequently, flow visualizations results and the comparison between Particle Image Velocimetry (PIV) measurements and numerical data are explained. The excellent agreement found makes the numerics to be reliable. A discussion of the results is given at the end of Section 2. Once numerical results are validated, heat transfer is computed numerically and explained in detail in Section 3. Finally, the main findings and conclusions drawn are described in Section 4.

2. Experimental and numerical results

2.1. Experimental setup

The schematic of the experimental setup is shown in Fig. 1 (a). Air compressed (A) to a high pressure in a deposit (B) flows at a specific rate Q and temperature T_{in}. The flowmeter (D) measures the flow rate at a specific aperture of the valve (C). The air flow emerges radially from an annular slot of height h = 10 mm and outer radius R_o = 50 mm, see Fig. 1(b), where a drawing for dimensions and additional details of diffuser, floor and chamber is depicted. The annular slot is on the top of a cylinder with height H_o = 212 mm (G). The inlet temperature T_{in} was controlled using a thermal bath (E). The air flow inside the cylinder was uniform thanks to an array of balls and a honeycomb. The cylinder and the floor are inside a chamber made with plexiglas to allow flow visualizations. This chamber has a volume of $L_3 \times L_1 \times L_1$ = 500 × 1000 × 1000 mm^3 with an ambient temperature, T_a, and L_3 being the floor to ceiling distance. The floor consists of a horizontal aluminium squared surface with a maximum length of 450 mm (L_2 = 50 mm) that is kept at a constant temperature T_f by means of a second thermal bath (H). This latter uses cool water through an array of parallel circular pipes inside the floor separated at a certain distance L_a = 100 mm. Experiments were conducted for temperature differences $\Delta T = T_a - T_{in}$ ranging from 1 to 3 K and the controlled T_f was varied from 290–300 K, with errors of ±0.1 K. Typically, T_a also varied in the range 296–303 K. This system simulates the condition of a diffuser coupled with radiant floor cooling. Flow rates Q employed were between 10 and 20 l/min, with errors of about ± 0.2 l/min. The flowmeter was previously calibrated with Laser Doppler Anemomentry (LDA) and PIV techniques. We used a Laskin nozzle generator (F) so the flow was visualized using oil micro-droplets seeded in the incoming air flow. The flow was illuminated by a continuous laser of 500 mW (I) that formed a sheet with the use of proper cylindrical and spherical lenses. One megapixel low-speed camera (H) up to 25 frames per second (fps) recorded digital images that were post-processed to obtain flow visualizations. One megapixel high-speed camera (J) up to 60,000 fps was also used for PIV measurements. The Matlab-based software DPIVSoft was employed to process the velocity field [48], using images with a time step of 1/250 fps and interrogation windows of 32 × 32 pixels. We captured and processed data for 400 instants of time to obtain PIV measurements.

The overall environment in the room where the experimental setup is located was conditioned at a constant room temperature. There were no heat loads inside the chamber of the experimental setup. Different radiative sources including short wave radiation and effects of reflecting surfaces are neglected, even in the numerical simulations (see below). In addition, we did not measure experimentally vertical temperature profiles. Finally, it is worth mentioning that steady state conditions were assumed when no fluctuations were observed in the laminar air flow by means of flow visualizations at least during 30 min. In this sense, signals of temperature were recorded at several sample points to ensure constant values for T_a, T_f and T_{in}.