Applied Thermal Engineering 87 (2015) 79–88

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research paper

Experimental and numerical study of a small-scale and low-velocity indoor diffuser for displacement ventilation: Isothermal floor

Alberto Fernández-Gutiérrez [a], Ignacio González-Prieto [a], Luis Parras [a], Paloma Gutiérrez-Castillo [b], José Manuel Cejudo-López [a], Carlos del Pino [a,*]

[a] Universidad de Málaga, Andalucía Tech, Escuela Técnica Superior de Ingeniería Industrial, Campus de Teatinos s/n, 29071, Málaga, Spain
[b] School of Mathematical and Statistical Sciences, Arizona State University, Tempe, AZ 85202, USA

HIGHLIGHTS

- Boundary conditions well posed in a small-scale experimental setup.
- Airflow patterns were recorded, finding good agreement with numerical simulations.
- Numerics allowed us to know convective heat transfer in the vicinity of the floor.
- Accurate correlations given: heat flux strongly depends on temperature difference.
- Accurate correlations given: heat flux weakly depends on flow rate.

ARTICLE INFO

Article history:
Received 2 July 2014
Accepted 1 December 2014
Available online 8 May 2015

Keywords:
Displacement flow diffuser
CFD
Thermal modeling

ABSTRACT

The accurate knowledge of the dynamics and heat transfer of an attached cool radial jet in a laminar regime, discharging onto an isothermal floor and undisturbed atmosphere, is crucial to better control the parameters and reach a comfortable environment. In the present study, we first analyze a simple geometry of a small-scale, indoor displacement ventilation diffuser, by means of flow visualizations, and the PIV technique. In addition, we have carried out axisymmetric numerical simulations that show excellent agreement with qualitative and quantitative experimental data. Finally, we provide correlations of the heat flux as a function of non-dimensional parameters, finding out that the Nusselt number has a strong dependence on the Grashof number rather than the Reynolds number in this configuration. This simple model is a relevantly simple tool for practical engineering purposes such as the conditioning of large public areas with displacement ventilation systems.

1. Introduction

Displacement ventilation consists of flowing large amount of air with a temperature slightly lower than the comfort one, and using the heat of people and equipments in the treated zone to convect the air to the ceiling. Displacement ventilation offers the utmost in occupant comfort because the temperature and ventilation are near the optimal working point (see Refs. [1,2], and the references therein for more information about displacement ventilation). The study of theoretical and typical cold air structures in displacement ventilation are of great interest to define optimal design procedures [3]. Displacement ventilation creates both a high temperature and ventilation effectiveness [4], but it is difficult to find a simple approach of this system [5]. The dynamic of this flow is connected to the so-called wall jet, that is, the resulting attached plane jet impacting and spreading radially onto a surface. This problem has been studied in a laminar and turbulent regime theoretically [6], and it is in agreement with experimental observations [7]. The flow behaves as a radial boundary layer far away from the axis. The wall jet has been compared experimentally in the turbulent regime in the past [8,9], and in a room taking into account bouyancy [10]. Recently, a new similarity structure for the boundary layer of a turbulent wall jet was presented [11]. The same configuration can be analyzed using the pressure distribution and the wall stresses on the surface, and considering the ideal flow and boundary layer equations [12,13]. This problem has also been widely studied in the literature adding the energy equation due to its industrial

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

http://dx.doi.org/10.1016/j.applthermaleng.2014.12.078
1359-4311/

applications. These applications include drying of paper, cooling of electronic equipments [14], or cooling turbomachinery elements [15], among others. The main work on the subject focused on the turbulent regime and the onset of primary or secondary vortices [16], together with the heat transfer, wall pressure distribution, and pressure loss [17]. These studies also focus on compressibility effects [18,19]. Generically, there are many difficulties in finding accurate turbulent models since the details of the experimental data are not well known. In addition, the thermal flow conditions may not be well posed in the geometry (see Ref. [20] and references therein). Furthermore, less information is given at moderate Reynolds numbers, beyond which the flow becomes unstable and even turbulent. A bidimensional flat jet has been also studied [21,22] with temperature variation and moderate Reynolds numbers, but no experiments were reported. This lack of knowledge in the laminar regime is the motivation for our work. The interest in this regime is to find out a first approximation of a low-velocity indoor diffuser for displacement ventilation.

On the other hand, there are many studies in the literature concerning heating, ventilation and air conditioning (HVAC) systems for large public buildings. The research in this field has been motivated by the desire to reduce energy consumption [23], while fulfilling the demand, together with the increment of comfort requirements [24], even for systems with radiant ceilings [25]. In these works, there is a natural stratification from the floor to the ceiling, and only the area near the inhabited floor (mixing zone) must be treated. To achieve this objective, the study of the flow in displacement ventilation systems by means of diffusers combined with refrigerated floors initiated [24]. Most of the thermal load should be removed by the refrigerated floor, while the diffusers will provide the ventilation load. Thus, the air is pushed by displacement ventilation at low velocities and set-point temperatures for cooling near the comfort in the treated zones are imposed. The air creates a cold bed near the floor. Once the air finds a thermal load (person or equipment), its temperature increases, so the air rises to the non treated zone. Recently, this process has been applied in large public areas [26,27]. In our work we shed some new light for small-scale experiments and thermal models in a laminar regime. We analyze a reduced-scale cylindrical diffuser to better understand the heat transfer process from an experimental and numerical points of view. Thus, the correlations for full-scale rooms with displacement ventilation systems [28], the non-uniformity of the indoor temperature field, the effect of adiabatic and non-adiabatic obstructions in the conditioned zone, or the presence of air perturbations among others [29], are outside the scope of this study. Therefore, this work is only focused on the exact solution for the simplest (laminar) case of a reduced-scale cylindrical diffuser with no external sources and non-occupancy loads and well posed boundary conditions. Consequently, the turbulent regime is neglected due to the fact that the laminar cool round jet does not impact with any heat source, hence the convection (mixed) zone is not created in the experimental setup.

One of the main challenges designing a HVAC system with displacement ventilation diffusers is to decide where to locate them, and this issue depends on the estimated load, among other factors. The exact position of these diffusers will improve the behavior of the whole system. For the sake of simplicity, the diffusers are installed equispaced, and little attention has been given to the general air distribution. Though this problem is very complex due to the nature of the flow (non steady, turbulent state with relatively high vertical temperature gradient in large areas), there are different turbulent models to achieve three-dimensional simulations successfully in a whole building [30]. However, these simulations are too time consuming, as the different boundaries and scales must be solved accurately [31], even by using nonlinear RANS models [32]. To avoid this problem, different authors have been trying to simplify the calculations to obtain a first approach for the air distribution. This will allow us the accurate modeling of convective heat transfer correlations in rooms in terms of dimensional or non-dimensional parameters (see Refs. [29] or [1] and the references therein). For this reason, the problem of finding an optimal spatial distribution of the diffuser on the floor based on simple models will be interesting for real and practical applications to evaluate and control indoor systems. This is the main objective of this paper. To that end, we simplify the problem, taking into account the following constraints: laminar regime, non-refrigerated floor and non-occupancy loads. Thus, the result obtained will be a first order approximation for practical design applications. In this study, we only pay attention to the isothermal case, being the effect of the radiant floor analyzed in other manuscript.

This paper is organized as follows. Firstly, a general description of the experimental setup and flow visualizations are included in section 2. The presentation of the laminar, steady and axisymmetric numerical results is given in section 3. The comparison between numerics and experimental data is shown in section 4. A simple laminar heat transfer model and its heat flux correlations are given in section 5. Finally, a summary of the main results is presented in section 6.

2. Experimental setup. flow visualization

We used a small-scale diffuser depicted in Fig. 1 (a) in order to obtain a simple model, as well as a set of well-posed boundary conditions. The cylinder was precisely manufactured in perspex with a outer diameter $2R_o = D = 100 \pm 0.06$ mm, thickness of 2.5 mm $\pm$ 0.05 mm, total height $H_0 = 212 \pm 0.05$ mm, and axial distance of the round slot $h = 10 \pm 0.05$ mm. This cylinder was inside a square cross-section chamber of one square meter ($L_1 \times L_1 = 1000^2$ mm^2), and a floor to ceiling distance of $FCD = 500$ mm. The cylinder was also isolated from the surrounding air fluctuations outside. Special care was taken at this point to avoid disturbances and keep the flow axisymmetric and steady. There was a gap around the perimeter of $L_2 = 50$ mm between the chamber and the floor to avoid the movement of the cool air. As a result, the outflow air moved towards to a low level of the conditioned room. To ensure the same temperature conditions in each test, both the qualitative and the quantitative measurements were taken simultaneously. The whole test section was made of perspex to allow optical techniques such as Mie-scattering [33] [see, for instance, Fig. 1 (b)], Laser Doppler Anemometry (LDA) or Particle Image Velocimetry (PIV). Flow visualizations were acquired using a standard one Megapixel digital video camera at 25 frames per second (fps). On the other hand, for the LDA measurements we used Dantec 1D equipment and for PIV measurements we used two 500 mW continuous laser and a high speed camera up to 60,000 fps, though only 250 fps were required. An aluminum structure was used to hold the experimental setup. The different sections (floor, cylinder, ceiling and walls) were accurately aligned with a digital inclinometer to within $\pm 0.1^\circ$.

The inlet flow rate, Q, and the inlet (T_{in}), ambient (T_a), floor (T_f) and ceiling (T_c) temperatures were controlled accurately by different equipments. To that end, we used digital and analogical flowmeters and two controlled heat exchangers for the inlet and floor temperatures.

Typical values of the flow rate Q were inside the range 10–19 l/min. The velocity inside the cylinder was uniform thanks to different layers of 6 mm diameter balls, stainless steel meshes of 0.25 mm^2 free space, and one honeycomb of 6 mm hexagons and 50 mm length. Taking into account the values of the geometry, the final velocity at the exit of the round slot is close to 0.1 m/s. This