Building and Environment xxx (2012) 1–12

Contents lists available at SciVerse ScienceDirect

Building and Environment

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

Properties of large-scale flow structures in an isothermal ventilated room

M. Körner [a,*], O. Shishkina [b], C. Wagner [b], A. Thess [a]

[a] Institute of Thermodynamics and Fluid Mechanics, Ilmenau University of Technology, P.O. Box 100565, 98684 Ilmenau, Germany
[b] DLR – Institute for Aerodynamics and Fluid Technology, Bunsenstrasse 10, 37073 Göttingen, Germany

ARTICLE INFO

Article history:
Received 25 July 2012
Received in revised form
16 October 2012
Accepted 17 October 2012

Keywords:
Isothermal ventilation
Turbulent large-scale flow structure
Auto-oscillation
Laser Doppler Velocimetry
Direct numerical simulation

ABSTRACT

In the present work experimental and numerical investigations of the large-scale structures of isothermal air flow in a highly simplified model room are reported and discussed. We compare the measured velocity distribution with direct numerical simulations (DNSs) for the reference Reynolds number $Re_{ref} = 2.4 \times 10^4$, which is based on the maximum inlet velocity and the height of the room. This comparison shows a high similarity concerning the flow structures. Furthermore, we investigate the dependence of the spatial and temporal behavior of flow structures on the Reynolds number from measurements covering a Reynolds number range of $1.0 \times 10^4 \leq Re \leq 7.0 \times 10^4$. The major finding is that there is a coherent oscillation of the large-scale flow structures, which depends on the Reynolds number. Our findings show that the frequencies of the oscillations are in a good agreement with an empirical model, which describes the auto-oscillation of two colliding planar jets with respect to the Reynolds number and the geometry relations of the inlets. Moreover, the present work indicates that the chosen flow geometry is well suited as a simplified model for problems of room ventilation and can serve as a base for forthcoming studies of non-isothermal cases.

1. Introduction

In view of the increasing significance of the energy efficiency of ventilation systems and the thermal comfort of persons inside a room within the past three decades, the knowledge of the spatial and temporal properties of large-scale air flows inside rooms has become more and more important. That knowledge can influence the complete design of rooms like passenger cabins or office rooms. In most cases the indoor air flow is a buoyancy- and momentum-driven flow with highly complex structures. To understand these complex non-isothermal air flows, it is necessary to fully understand the behavior of the isothermal air flow. Thus, we experimentally and numerically investigated the properties of the large-scale flow structures in a ventilated model room for the isothermal case in the present paper.

First experimental and numerical investigations of isothermal ventilation of rooms have been reported by Nielsen et al. [1]. In this investigation a simple model room with rectangular cross-sections and one inlet and one outlet, the so called Nielsen room, has been used to study the fundamental behavior of the isothermal air flow. Nielsen reported that the flow structures have a high dependency on the geometrical dimensions of the room and the inlets but also on the inlet flow velocity. Nielsen's model room has been investigated in several further investigations, for example by Kato et al. [2], Gosman et al. [3], Mora et al. [4] or Mushatet [5]. Nielson's findings regarding the ventilation of a model room are often used to verify numerical simulation like Mora et al. [4] did. Other investigations related to this model room concerned about fundamental information of the ventilation efficiency [2] or to get further information regarding the dependency of the flow on the boundary conditions [3,5].

In the present work, the geometry of the model room to be investigated experimentally and numerically relates in general as well to Nielsen's work and is shown in Fig. 1a.

The model room we use could be understood as an extension of the Nielsen room to a more complex geometry. We also consider a box with rectangular cross section. But in contrast to Nielsen's room our model room contains four rectangular flow obstacles near the bottom which extend over the full depth of the room. The ventilation is realized through two opposed horizontal inlets near the ceiling of the room. The air exits the room through two outlets at the bottom. The inlets and outlets also extend over the full depth of the room. So, the room geometry is in general and in contrast to Nielsen's geometry more related to a passenger cabin, e.g. an aircraft cabin.

* Corresponding author. Tel.: +49 3677 69 4402.
E-mail addresses: max.koerner@tu-ilmenau.de (M. Körner), olga.shishkina@dlr.de (O. Shishkina), claus.wagner@dlr.de (C. Wagner), thess@tu-ilmenau.de (A. Thess).

0360-1323/$ – see front matter © 2012 Published by Elsevier Ltd.
http://dx.doi.org/10.1016/j.buildenv.2012.10.011

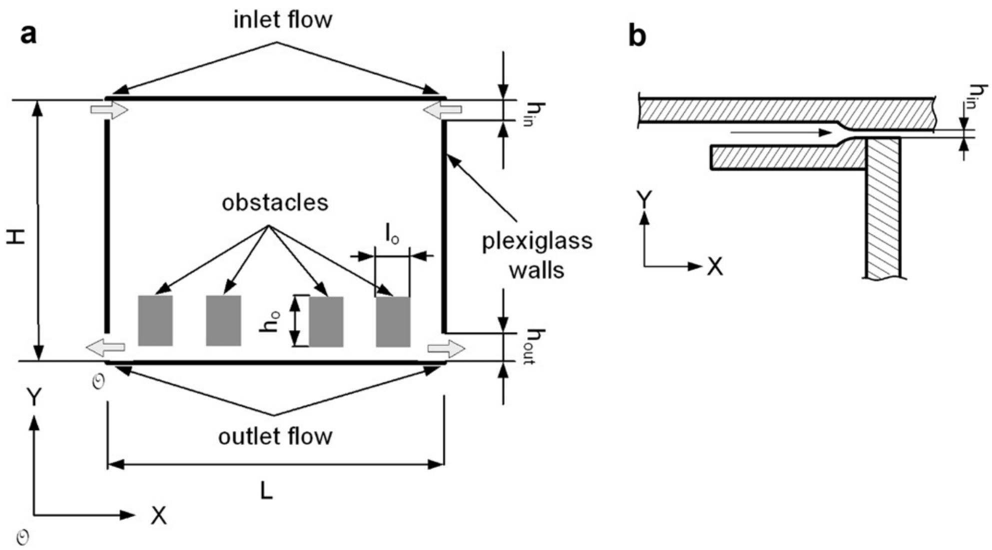

Fig. 1. a) Cross-section of the model room at $z/D = 0.5$; two inlets are at the top of the room and two outlets at the bottom; four obstacles are arranged at the bottom which extend through the whole depth (normal to the cross-section) of the room; b) Detail of the inlet at $x/L = 0$ and $y/H = 1$.

With respect to this room geometry, we are interested in answering particularly the following questions: What are the spatial properties of the large-scale structure of the flow? How well can a direct numerical simulation (DNS) in a box with a smaller depth than the experimental one reproduce the large-scale flow structures? Is there a spatial and temporal dependency of the behavior of the large-scale flow structure on the Reynolds number Re?

Here, the Reynolds number is defined as

$$\mathrm{Re} = \frac{u_{\mathrm{in}} \cdot H}{\nu} \tag{1}$$

with u_{in} as the maximum velocity of the inlet flow, H as the height of the room and ν as the kinematic viscosity of dry air.

The outline of the present work is as follows. In Section 2 we describe the experimental setup, the used measurement setup and the data processing. In Section 3, we describe the numerical domain and the computational algorithms of the DNS. Section 4 contains the description and the discussion of the results of our investigation. We finish the present work with a conclusion in Section 5.

2. Experimental setup

Fig. 1a depicts a sketch of the cross-section of the investigated model room. The model room is equipped with two inlets in the side walls and close to the ceiling of the room. The shape of the inlets is shown in Fig. 1b. Two outlets are arranged in the same side walls like the inlets but close to the bottom of the room. Near the bottom of the room are four rectangular flow obstacles. This geometry is extended over the full depth D. The model room is made of Plexiglas and the flow obstacles are made of aluminum.

We define the origin of the coordinate system in the left, lower corner with the x-direction along the length L, the y-direction along the height H and the z-direction along the depth D. The height of the model room is $H = 300$ mm. Other dimensions describing the model room are $\Gamma_{xy} = L/H = 1.33$; $\Gamma_{zy} = D/H = 1.67$; $h_{\mathrm{in}}/H = 0.0067$ and $h_{\mathrm{out}}/H = 0.05$, as shown in Fig. 1. The inlets and outlets have the same depth as the model room ($d_{\mathrm{in}}/D = 1$; $d_{\mathrm{out}}/D = 1$). So, according to the findings of Nielsen [1], we can assume that the large-scale flow is nearly 2-dimensional. To abide this assumption, it is necessary to generate an inlet flow with a homogeneous velocity distribution along the z-direction. This is realized by a nozzle, as shown in Fig. 1b, and prechambers in front of the inlets. The inlet flow is generated with air from a pressure vessel with an adjustable outlet pressure for adjusting the inlet velocity.

The obstacles within the room have a cross-section of $h_o/H = 0.2$ and $l_o/L = 0.1$. The depth of the obstacles is almost of the same size as the depth of the model room. Only a small gap of $d_o/D = 0.01$ at each side of the model room exists. The obstacles are arranged at the positions $x/L = [0.1, 0.3, 0.6, 0.8]$ related to the origin and the left side of the obstacles in Fig. 1a. The distance from the bottom is $h/H = 0.05$.

We measure the horizontal and vertical velocity component u and v in the central x–y-cross-section with a 2D FiberFlow Laser Doppler Velocimetry (LDV) System from Dantec Dynamics with a 3D traverse system, as shown in Fig. 2. According to the work of Tridimas et al. [6], we estimated the radius of the beam waist of a pair of idealized Gaussian laser beams. From that we calculated the maximum width Δx_{mv} and the maximum length Δz_{mv} of the

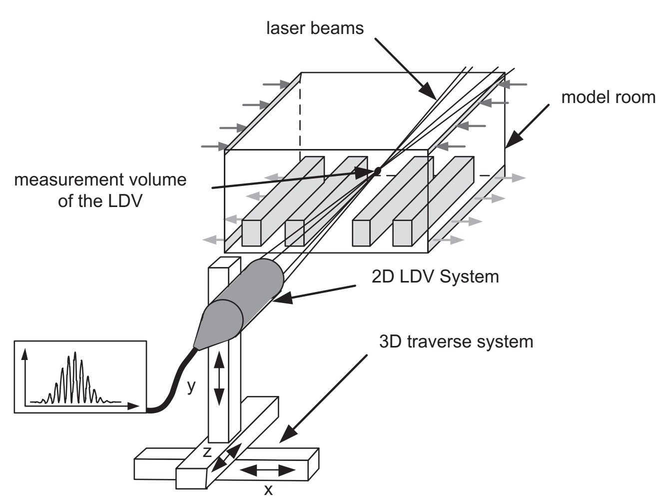

Fig. 2. Schematic of the measurement setup with the 2D Laser Doppler Velocimeter (LDV) on a 3D traverse system in front of the model room.

Please cite this article in press as: Körner M, et al., Properties of large-scale flow structures in an isothermal ventilated room, Building and Environment (2012), http://dx.doi.org/10.1016/j.buildenv.2012.10.011