Energy and Buildings 127 (2016) 907–919

Contents lists available at ScienceDirect

Energy and Buildings

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

Optimized performance of displacement ventilation aided with chair fans for comfort and indoor air quality

Walid Abou Hweij, Nesreen Ghaddar*, Kamel Ghali, Carine Habchi

Mechanical Engineering Department, Faculty of Engineering and Architecture, American University of Beirut, P. O. Box 11-0236, Beirut, 1107-2020, Lebanon

ARTICLE INFO

Article history:
Received 22 April 2016
Received in revised form 14 June 2016
Accepted 16 June 2016
Available online 16 June 2016

Keywords:
Displacement ventilation
Chair fans
Thermal comfort
Indoor air quality
Energy savings

ABSTRACT

This study optimizes the performance of displacement ventilation (DV) system aided with chair fans (CF) for providing acceptable thermal comfort and indoor air quality (IAQ) in office space. A 3-D computational fluid dynamics (CFD) model coupled with a bio-heat model was used to predict the airflow, temperature, and CO_2 concentrations fields as well as the occupant segmental skin temperatures for local and overall thermal sensation and comfort evaluation. The CFD model results were validated experimentally in a DV conditioned space using a thermal manikin seated on a chair equipped with fans.

Simulations were performed using the validated CFD model to determine the fans optimal height from the floor. For a pollution source located at 1.0 m from the floor, it is recommended to place the fans at 30.9 cm above the floor at fans total flow rate of 12 L/s while for a pollution source located at 0.3 m from the floor, the optimal height of fans is at 30.1 cm above the floor at fans total flow rate of 12 L/s. Compared to stand alone DV system, energy savings were 20.6% and 11.6% for fans' optimal heights of the sources placed at 1.0 m and 0.3 m, respectively.

1. Introduction

Acceptable thermal comfort and good indoor air quality (IAQ) are the main requirements to be provided by ventilation systems in open space office buildings. Displacement ventilation (DV) system supplies 100% fresh and cool air at low velocity near the floor level from a wall mounted diffuser relying on heat sources that push the resulting warm and polluted air by buoyancy to the upper room level to be exhausted; thus creating temperature stratification within the space [1]. This system requires the conditioning of the 100% fresh air supply stream which is energy intensive compared to mixed streams used in conventional. Additionally, the restrictions on the supply air temperature (not to be less than 18 °C) and flow velocities (not to be more than 0.2 m/s) [2,3], hindered the DV system from removing high internal loads. Thus, DV system might not be cost effective in hot and humid climate conditions. Furthermore, Brohus et al. [3,4] reported poor performance when a passive contaminated source existed at low level.

Several studies proposed different approaches to improve the performance of DV system. One approach was to use chair-mounted fans of an office occupant firstly proposed by Sun et al. [5–10]. Sun et al. [5,7] proposed the use of four computer fans placed at the corners of the chair. They surveyed participants about their thermal comfort while changing speed of the fans for different room air temperatures. Their study suggested the appropriate speed of fans that provided thermal comfort for occupants for the various room temperatures. Moreover, Sun et al. [7,8] investigated experimentally the effect of chair fans (CF) on an ambient passive contaminant represented by CO_2 tracer gas. The study showed that for the proposed configuration of fans (at the level of thighs), the fans increased the concentration of CO_2 at the occupant's breathing zone. Additionally, they found that the use of this novel system resulted in a reduction in energy consumption up to 19% [7]. Habchi et al. [11] studied the performance of DV system assisted by chair fans (DV + CF) in reducing particles transmission. They reported that the use of fans caused a reduction in inhalation effectiveness when compared to a standalone DV system. In addition, they reported that the novel system provided thermal comfort when assessed using predicted mean vote (PMV) model. Evidently, using CF in aiding DV system had noticeable effects on thermal comfort and particles transmission while reducing energy consumption.

All the previous studies focused on the use of CF at a fixed level without consideration of the effect of the height while varying fan flow rate on thermal comfort and IAQ. Recent study done by El-Fil et al. [12] showed noticeable improvements in thermal comfort and IAQ as a result of variations in height and flow rate of CF while

* Corresponding author.
E-mail address: farah@aub.edu.lb (N. Ghaddar).

http://dx.doi.org/10.1016/j.enbuild.2016.06.052
0378-7788/

Nomenclature

Symbols

C_B	CO_2 concentration of the breathing zone (ppm)
C_{Fr}	CO_2 concentration of the fresh air (ppm)
C_r	CO_2 concentration of the recirculated air (ppm)
CF	Chair fans
CFD	Computational fluid dynamics
CPV	Ceiling personalized ventilation
DV	Displacement ventilation
DV + CF	Displacement ventilation assisted by chair fans
H_{fs}	Height of fans relative to floor (cm)
H_s	Height of source relative to floor (cm)
IAQ	Indoor air quality
LTC	Local thermal comfort
LTS	Local thermal sensation
MV	Mixed ventilation
PMV	Predicted mean vote
PPM	Parts per million
$\dot{Q}$	Total flow rate of fans (L/s)
RH	Relative humidity (%)
TC	Overall thermal comfort
TS	Overall thermal sensation
y	First inflation layer thickness (mm)
y^+	Dimensionless wall thickness (−)

Greek symbols

ε_v	Ventilation effectiveness (%)
ρ	Air density (kg/m^3)
μ	Air dynamic viscosity (Pa s)
τ_w	Wall shear stress (Pa)

aiding ceiling personalized ventilation system (CPV). In DV system, varying the height of the fans above the floor while varying their flow rate may affect the thermal comfort. Moreover, the use of CF at different heights and flow rates may affect the breathing air quality when passive contaminants are placed at different heights above the floor. To our knowledge, the effect of fans height above the floor and their flow rate on breathing air quality and comfort at different passive contaminant height above the floor has not been investigated in the literature.

Hence, this study will investigate the performance of (DV + CF) system with the purpose of optimizing the height of fans above the floor for different passive contaminant height. A computational fluid dynamics (CFD) model of the DV-conditioned space with chair fans is developed. The CFD model will be validated experimentally using thermal manikin to compare the segmental surface temperature and CO_2 contaminant concentration in the breathing zone at a specified source height. A parametric study will follow to assess the system performance in providing acceptable thermal comfort and good IAQ for an office occupant at low energy consumption.

2. Methodology

Fig. 1(a) shows a schematic diagram of (DV + CF) system. The supply diffuser supplies fresh air into the room, the thermal manikin represents an office occupant, the chair mounted fans, placed on each side, blow air upwards, the CO_2 tracer gas represents passive contaminants in the room, and the exhaust diffuser extracts contaminated air from the space. Fig. 1(b) and (c) shows in details the configuration of chair mounted-fans. The fans can move vertically up and down resulting in a controlled height $\left(H_{fs}\right)$ while supplying total flow rate $\left(\dot{Q}\right)$ equally distributed over the four fans.

To study the effect of various parameters; including height of fans above the floor, flow rate of fans, and passive contaminant source height above the floor, on the performance of (DV + CF), a 3-D CFD model is used to investigate the ability of the (DV + CF) system with the controllable height in providing comfortable environment while aiding the thermal plumes in delivering a relatively clean air at the breathing level of the occupants. Experiments are conducted to validate the findings predicted by the computational model. The predicted CO_2 concentration fields along with the segmental surface temperatures of the modeled occupant are then compared with the experimental measurements. Following the experimental validation, the CFD model is used to run a parametric study on a typical office space with specified load to assess and optimize the performance of the proposed system (DV + CF) under the effect of various parameters while providing low energy consumption.

2.1. CFD model

The complex interactions resulting from the upward jets of fans and the rising thermal plumes should be accurately predicted to assess their behavior on the thermal and CO_2 concentration fields. Similar studies done by Makhoul et al. [13] resulted in accurate prediction of the airflow and thermal patterns along with the CO_2 concentration fields. Thus, CFD modeling would be used to investigate the effectiveness of the proposed (DV + CF) system under different configurations.

2.1.1. Airflow modeling

The commercial software ANSYS© Fluent was selected for assessing the flow fields of indoor environment as shown in Fig. 2. Accurate prediction of the airflow, thermal, and CO_2 concentration fields are based on proper modeling of flow physics including turbulence models, buoyancy effects, and boundary layers near the surfaces [14]. Proper controlling of the grid resolution at some surfaces is crucial to capture perfectly the shear-layer entrainment process and the different thermal plumes as well as the fluid/thermal boundary layers around the occupant [13]. The flow behavior at the boundary layer is accurately predicted using inflation layers. The selection of the inflation layers is based on first layer thickness y such that the dimensionless number y^+ ranges between 0.8 and 4, in order to an accurately resolve the viscous sub-layer as reported in ANSYS online manual [15]. Accordingly, the first layer thickness based on a selected y^+ value is estimated as follows:

$$y = \frac{y^+\mu}{\rho\mu_\tau} \tag{1a}$$

where ρ and μ are the air density and dynamic viscosity and μ_τ the friction velocity is given by:

$$\mu_\tau = \sqrt{\frac{\tau_w}{\rho}} \tag{1b}$$

where τ_w is the wall shear stress.

The boundary faces were set to different element sizes using tetrahedral unstructured grid. A grid independence test was performed. The mesh is considered independent when the maximum relative error between two consecutive grid sizes is less than 5%. Initially, a face size of 2 cm and 8 cm were assigned to the surfaces of the manikin and the walls respectively resulting in 766,144 elements and 184,707 grid nodes. The face size was increased where the temperature, velocity and concentration fields at certain locations were compared with the previous grid test. This was repeated until grid independence was assured. A total number of 3,516,946 elements and 876,576 grid nodes were selected to accurately predict the flow behavior with less computational cost as illustrated in Table 1. Fig. 3 represents (a) the generated grid for the computational domain and (b) generated grid for the manikin.