Building and Environment 130 (2018) 14–26

Contents lists available at ScienceDirect

Building and Environment

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

Experimental and numerical investigations of indoor air movement distribution with an office ceiling fan

Wenhua Chen[b], Shichao Liu[a,*], Yunfei Gao[c,d], Hui Zhang[c], Edward Arens[c], Lei Zhao[b], Junjie Liu[b]

[a] Department of Civil and Environmental Engineering, Worcester Polytechnic Institute, Worcester, MA 01609, USA
[b] Tianjin Key Laboratory of Indoor Air Environmental Quality Control, School of Environmental Science and Engineering, Tianjin University, Tianjin, China
[c] Center for the Built Environment, University of California, Berkeley, CA, USA
[d] School of Architecture and Urban Planning, Guangdong University of Technology, Guangzhou, China

ARTICLE INFO

Keywords:
Ceiling fan
Blade geometry
Rotational speed
Air movement distribution
Moving reference frame
Thermal comfort

ABSTRACT

Ceiling fans provide cooling to indoor occupants and improve their thermal comfort in warm environments at very low energy consumption. Understanding indoor air distribution associated with ceiling fans helps designs when ceiling fans are used. In this study, we systematically investigate the air movement distribution in an unoccupied office room installed with a ceiling fan, as influenced by (1) fan rotational speed, (2) fan blade geometry, (3) ceiling-to-fan depth, and (4) ceiling height. We both measured and simulated air speeds at four heights in the occupied zone according to *ANSI/ASHRAE/IES Standard 55 (2013)* for seated and standing occupants. CFD predictions were validated by experimental results. In general, numerical results show that for an unoccupied space, the fan blade geometry, ceiling-to-fan depth, and ceiling height only influence air speed profiles within a cylindrical zone directly under a ceiling fan whose diameter is identical to that of the ceiling fan. However, the average speeds within the cylindrical zone at each height are very similar (< 10% in difference) for the different blade shapes studied, indicating a minor influence of blade geometries on occupants' perception of the thermal environment. The results also indicate that the velocity profile remains similar in the main jet zone (the tapered high-velocity zone under the fan blade) for various rotational speeds. The jet impingement on the floor creates radial airflow at the ankle level (0.1 m) across the room, which is not the most effective airflow distribution for cooling occupants.

1. Introduction

Energy constraints encourage the use of energy-effective electrical appliances such as ceiling fans to achieve indoor thermal comfort, especially in developing countries (e.g., China and India) and regions with mild and warm climates. Ceiling fans accounted for approximately 6% of residential electricity consumption in India in 2000, which might increase to 9% in 2020 [1].

Elevated air speeds from ceiling fans can offset the need for low thermostat cooling set-points, and provide occupants with enhanced thermal comfort at a lower energy consumption. *ANSI/ASHRAE/IES Standard 55 (2013)* [2,3] recommends an elevated air movement method to maintain thermal comfort in the occupied zone at increased indoor temperature. Airflow from a ceiling fan with a speed between 0.5 m/s and 1.0 m/s compensates for approximately 3 K indoor temperature increase [4] or even more [5]. Energy simulations suggest a saving of 15% residential cooling energy by using ceiling fans and a thermostat set-up of only 1.1 K [6], saving between 17% and 48% for 400 Florida households [7]. Additional simulations show that in commercial office buildings, a 1 K setpoint extension is associated with about 10% HVAC energy savings in most types of climate [8].

1.1. Ceiling fan evaluation approaches

The performance of ceiling fans is often evaluated from the perspective of energy consumption for created airflow rates [9–11]. Overall, a ceiling fan that provides a certain airflow rate at a lower power input is rated with a higher efficiency. The American Energy Star program defines the minimum efficacy levels for certified ceiling fans. For instance, fans at a low speed must have a minimum airflow rate of 2124 m^3/hr and an efficiency of 263 m^3/hr/W [10] A nice summary of the ceiling fan energy efficiency evaluation methods is presented by de la Rue du Can et al. [1]. However, such performance evaluation does not consider air movement distribution within the room, which is the feature that affects occupants' thermal comfort.

Schiavon and Melikov [12] developed a cooling-fan efficiency (CFE)

* Corresponding author.
E-mail address: sliu8@wpi.edu (S. Liu).

https://doi.org/10.1016/j.buildenv.2017.12.016
Received 1 October 2017; Received in revised form 30 November 2017; Accepted 15 December 2017
Available online 18 December 2017
0360-1323/

index to assess the performance of cooling fans, including ceiling fans, by considering the reduction of equivalent temperature that is caused by elevated air movement when compared to an environment with still air. The CFE is defined as the ratio of cooling effect to fan power. This method combines the factors of both the energy saving and thermal comfort rather than pure airflow capacity. Based on the comfort with elevated air movement described in the *ANSI/ASHRAE/IES Standard 55 (2013)* [2], the CFE index relies on the room air distribution to provide thermal comfort at any given point.

A new ASHRAE Standard 216 (ceiling fan method of testing) is currently under development to quantify and to predict airflow under ceiling fans. To evaluate fan performance, the existing indexes focusing on energy efficiency and airflow volume might not be enough. An index is also needed that links fan speed to the distribution of air movement in the occupied zone, since that determines occupant thermal comfort. This second index is included in the goals of ASHRAE Standard 216. A final goal of the Standard is a design tool to provide guidance for designers when ceiling fans are applied. For the latter two goals, understanding air movement distribution in a room is a key component. This team performed activities to support the ASHRAE Standard 216 effort. First, detailed laboratory measurements of occupied zone air speeds, with and without furniture, have been taken under a single ceiling fan. The detailed measurements profiles are described [13].

1.2. Factors affecting air flow distribution by ceiling fans

Most fan studies have focused on the air flow volume induced by ceiling fans, but not on the room air movement distribution, and particularly not on the distribution in the occupied zone. Laboratory [14,15] and field [16] studies compared energy efficiency index and indoor airflow distributions from different fan speeds and blade diameters. The studies indicated that the vertical temperature difference decreases with increasing fan speed, and that wider fan blades increase airflow coverage with increased energy efficiency [16], and that fans with a larger diameter and lower rotation speed reduce noise [17]. There are a few studies that examine the effect of air speed on bacterial removal [18–21], all finding that increasing speed results in higher disinfection efficacy. These studies did not examine the air movement distribution in the occupied space.

There has been considerable effort to improve the design of the fan blades in order to increase the flow volume, uniformity along the fan radius, and to increase the air movement coverage area. Adeeb et al. [22] focused on the number of blades and found that increasing the number of blades resulted in a higher flow volume. Afaq et al. [23] measured the effect of rake angles on the flow volume and found that a 6° upward rake angle (fan blades tilted above the horizontal level) provided the highest flow volume. Jain et al. [24] found that introducing winglets and spikes on the blade tip increased flow volume.

Volk [25] designed aerodynamic attachments as auxiliary blade attachments wedged onto the trailing edges of the main blades of conventional ceiling fans. Bird [26] designed a positive twist adjacent to the rotor end of the blade, so that blade pitch increases from a tip end of the blade to the rotor end of the blade. Sonne & Parker [16] developed twisted and tapered blades with airfoil cross-section. The twisted, tapered blades increase airfoil efficiency by reducing energy lost to wingtip turbulence and flow separation. Parker published several patents [27–29] about the blade shape, providing the basis for the “Gossamer Wind” fan. Schimidt and Patterson [9] evaluated the power consumption/airflow power of ceiling fans with an axial flux brushless DC motor. The fan blades were reshaped aerodynamically to provide a more constant air velocity across the area below the blades.

Additionally, laboratory measurements [30] indicated that a ceiling surface began to reduce fan flow volume only when the distance between the ceiling and the fan (diameter is 1.4 m) is 0.4 m or less.

1.3. CFD models

Numerical simulation using CFD has been successfully applied in ventilation and indoor air distribution research [31–33]. All the reviewed work used Reynolds-averaged Navier-Stokes equation (RANS) models because of their less computational requirements compared to Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) [34,35]. Momoi et al. [30] compared the standard k-ε turbulence model and the Reynolds stress model (RSM) in a study of turbulence models used in ceiling fan simulations. The comparisons indicated unremarkable differences between the two models.

Because of the complexity of ceiling fan’ geometries, it is generally cost-prohibitive to directly model a rotating ceiling fan using the moving mesh method [36,37]. Instead, the body force model (also called momentum method) and multiple reference frame (MRF) method are often applied [36,38]. Bassiouny et al. [39] fitted the previous measured data [24] as Dirichlet-type boundary conditions based on the body force model, and predicted the airflow induced by a ceiling fan using the standard k-ε model. Zhu et al. [19] used MRF together with the realizable k-ε model to examine the ceiling fan-driven flow in an environmental chamber. Contrary to these, Babich et al. [38] concluded that SST k-ω model is better than the standard k-ε and realizable k-ε models when applying the body force model to ceiling fan simulations. However, the conclusion was drawn when the ceiling fan was simulated by a body force model rather than MRF.

We identified the following limitations of previous related studies. First, the possible effects of fan blade geometries on airflow patterns have not been thoroughly investigated. The challenges of considering fan blade geometries in CFD simulations entail investigations into how much air distribution varies with different fan geometries. Second, even if previous studies have recognized that using ceiling fans is an energy-efficient solution for thermal comfort via air movement elevation, many factors, such as fan rotational speed and installation height, could influence air distribution and thermal environment in the occupied zone. Insufficient guidance about these factors inhibits designers and practitioners from applying ceiling fans properly in practice. Finally, turbulent jets usually display the properties of self-similarity in the self-preservation region (further down from the initial development region close to the fan blades) where the flow profiles are self-similar [40]. We hypothesize that similar self-similarity might exist for the primary airflow induced by a ceiling fan. Nevertheless, we could not find such an investigation in the literature.

1.4. Objectives

This paper aims to provide fundamental knowledge on how fan properties and room configuration affect the air movement distribution in an office room. In particular, the present study systematically investigates the influences of (1) ceiling fan rotational speed, (2) fan blade geometry, (3) distance between ceiling and fan, and (4) ceiling height on air distribution in an unoccupied office, using computational fluid dynamics (CFD) simulations. The CFD simulations were first validated by laboratory experiments and then applied to investigate these factors.

2. Methodology

2.1. Baseline experimental data

Simulations in this study are based on a previous experimental study of an office room conducted at the Center for the Built Environment (CBE) at University of California, Berkeley [13]. The room had geometries of 5.5 (X) × 5.5 (Z) × 2.5 (Y) m, as shown in Fig. 1a. A ceiling fan (Haiku 60, Big Ass Fans, Inc.), 1.5 m in diameter, was installed at 0.2 m below the ceiling surface. Fig. 1b is a snapshot of the fan used in the experiment. Smoke visualization was used to ensure that the room