Building and Environment 123 (2017) 37–49

Contents lists available at ScienceDirect

Building and Environment

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

Transient three-dimensional CFD modelling of ceiling fans

Francesco Babich [a,*], Malcolm Cook [a], Dennis Loveday [a], Rajan Rawal [b], Yash Shukla [b]

[a] *School of Civil and Building Engineering, Loughborough University, Loughborough, UK*
[b] *CEPT University, Ahmedabad, India*

ARTICLE INFO

Article history:
Received 15 April 2017
Received in revised form
15 June 2017
Accepted 21 June 2017
Available online 23 June 2017

Keywords:
Ceiling fan
Thermal comfort
CFD validation
Turbulence modelling
India
Environmental chamber

ABSTRACT

Ceiling fans have been used for decades as a means of providing thermal comfort in tropical countries such as India. However, recent years have witnessed a significant increase in the use of air conditioning as a means to achieve comfort, and therefore in the total energy consumption and related CO_2 emissions. Ceiling fans are still viable options to limit use of air conditioners or in combination with air conditioners without compromising on thermal comfort and still achieving energy savings. Ceiling fans generate non-uniform velocity profiles, and therefore relatively non-uniform thermal environments, whose characteristics may be tough to analyse with simple modelling methods. This issue can be investigated using CFD. However, to date, there are few works on ceiling fans, CFD and thermal comfort. More accurate models are therefore required to predict their performance. The research presented in this paper aimed to develop and validate a three-dimensional transient implicit CFD model of a typical ceiling fan available in India by comparing simulation results obtained using different URANS turbulence models with measured data collected in controlled environment. The results highlight that this ceiling fan model is able to replicate the predominant characteristics of the air flow generated by the fan such as the meandering plume and the local fine free shear layers. The best results are achieved when the SST k-ω turbulence model is used, with 83% of the simulated values being within the error bars of the respective measured value.

1. Introduction

Ceiling fans have been used for decades as a means of providing thermal comfort in tropical climates. In Indian residences, ceiling fans are as common as electric light bulbs, being present in almost every habitable space. They are part of most of Indian residences, and they are widely used in both old and more recent buildings [1,2]. In the event of growing economy and rising percentage of population which can afford purchase and operation of air conditioner for higher want of thermal comfort India has experienced rise in sales of air conditioners [3]. Cities such as Delhi, Mumbai and Kolkata, having Cooling degree days in range of 3000 and 3500, are replacing fans with air conditioners despite possibility of use of fan during certain part of year. It is important to note that most air conditioners have low-efficiency and use high-GWP refrigerants [4]. The electricity demand for space cooling comprises up to 60% of summer peak load in large cities such as Delhi [5]. Unless the use of energy-intensive air conditioning is limited only to periods of extremely hot weather, then overall Indian energy consumption and related CO_2 emissions will significantly increase, leading to severe implications for the global climate and also challenging the reliability of the Indian electricity grid.

Air conditioning usually provides uniform thermal environmental conditions, therefore designers can predict with confidence its performance using traditional thermal comfort models [6]. On the other hand, ceiling fans generate non-uniform velocity profiles, and therefore relatively non-uniform thermal environments. This does not imply a lower thermal comfort for the occupants, but could possibly lead to more thermally comfortable environments with lower energy cost due to air velocity, as research on alliesthesia suggests [7]. The positive effect of air movement on thermal comfort in warm and hot conditions has also been included in international standard such as the current version of ASHRAE 55 [8]: for operative temperature above 25.5 °C, the air speed limits have been raised to 1.2 m/s with occupant control and 0.8 m/s without occupant control. However, accurate models of the ceiling fans are required to predict their ability to generate air velocity and their effect on occupant comfort using newer and more advanced

* Corresponding author.
E-mail address: f.babich@lboro.ac.uk (F. Babich).

http://dx.doi.org/10.1016/j.buildenv.2017.06.039

thermal comfort models such as the IESD-Fiala model [9,10] coupled with a commercial CFD software [11]. Traditional thermal comfort models, namely the PMV-PPD model developed by Fanger [6] and the more recent adaptive approach [12,13], are indeed not suitable to investigate thermal comfort in non-uniform thermal environments. The former is applicable only to uniform steady-state conditions, while the adaptive model uses only the outdoor temperature as input parameter, and therefore it cannot be used to study the effect of a specific device such as a fan on thermal comfort.

The IESD-Fiala model is a model of human thermal physiology and comfort that predicts passive [9] and active [10] reactions of a human body in response to certain environmental conditions. It has been successfully coupled in real-time with a commercial CFD software [11], where real-time means that at each time step of a transient simulation there is an exchange of information between the two parts of the system. Thus, this coupled system can accommodate transient and asymmetrical environments [14], and can therefore be used to model the usage of ceiling fans, for instance identifying the best location for their installation. In the literature, there are other advanced models such as the model developed at UCB by Huizenga [15], but their capabilities are lower. Like the IESD-Fiala model, the UCB model is based on the multi-node model developed by Stolwijk for aerospace applications [16,17], and it has been used together with CFD software [18–20]. However, in these studies there is only a manual coupling, with no real-time automated procedure.

To date, there are few existing reports on the use of CFD to model ceiling fans. Bassiouny et al. [21] implemented simple 2D models. Thus, the truly three-dimensional behaviour of air flow generated by a ceiling fan is not captured. Momoi et al. [22,23] developed too complex modelling approaches that are limited in their application because many measurements are needed due to the required input data. Adeeb et al. [24] focused on the effect of the different numbers of blades, but their work did not provide sufficient information to build a 3D model of a ceiling fan in CFD to be used for thermal comfort studies within the Fiala-IESD and CFD coupled system. To do so, the most important thing is accurate modelling of the environment around the occupants, but keeping the CFD model as simple as possible for two reasons: reducing the time and computational power required to achieve a converged solution, and avoiding potential sources of error due to the use of an unnecessary complex model. For instance, the use of a moving mesh used by Zhu et al. [25] to study the effect of the ceiling fans on air mixing and UR-UVGI disinfection efficacy [25] would not guarantee more accurate thermal comfort predictions, but would certainly increase the computational time, especially when transient and steady-state simulations are used.

Thus, the research presented in this paper aimed to develop and validate a three-dimensional transient CFD model of a typical ceiling fan predominantly used in India by comparing simulation results and measured data. This model combines accuracy with efficient computation, and can be used for accurate thermal comfort studies.

This paper is divided in to five parts: (1) Introduction (2) Methods, this has two sub sections one deals with experiment and second modelling (3) Results from experiment and modelling (4) Discussion (5) Summary, including key findings and limitation of work.

2. Methods

Detailed measurements of the air movement generated by a typical Indian ceiling fan have been collected in an environmental chamber in controlled environment, and the same experimental set-up has then been modelled using a commercial CFD program, ANSYS CFX [26]. In this model, the fan is modelled as a momentum source that is specified by radial components and applied to a thin cylindrical sub-domain of the CFD model which has the same diameter as the fan used in chamber experiment. Measured values and computer-generated predictions have been compared analysed, and the reasons for any deviation discussed.

2.1. Experimental set-up

Environmental chambers have been extensively used to collect data for validating CFD models [27]. Within an environmental chamber, most of the variables can be controlled, and the state of the uncontrolled variables ought to be accurately determined. Thus, the number of required assumptions due to the lack of measurements or better information, and therefore the number of potential sources of uncertainties, are minimised.

For this research, an environmental chamber located at CEPT University, Ahmedabad, in India was used to generate validation data (see Fig. 1). Chamber is located in basement with only one side exposed to outdoor environment. Internal (thickness 200 mm) and external (thickness 350 mm) walls have U-value of 0.29 and 0.28 W/m^{2}K, respectively. Roof, false ceiling, and floor have U-value of 0.29, 0.55 and 2.80 W/m^{2}K, respectively. External window facing West direction has U-value of 1.79 W/m^{2}K. At no point in time direct solar radiation hit this window as it is well protected outside. The mechanical ventilation system comprises four supply grilles and two return grilles, which are located in the ceiling. The chamber ventilation system was turned off during any experiment, and the diffusers were sealed during both the second and third repetition of the measurements. This did not produce any significant difference in the measured values, but allowed to more accurately set the boundary conditions in CFD. Moreover, since the objective was to obtain data of velocity profiles under uniform and stable environmental conditions, the experiment was conducted once the chamber stabilized at constant air and surface temperatures. The stability of chamber was determined based on less than 0.1 °C change in air and surface temperature for more than 12 h. A typical 3-blade 1200 mm sweep Indian ceiling fan [28] with a 4-step regulator was installed on the ceiling of the chamber. The diameter of central part is 190 mm and the length of the blade is 500 mm. Other than measuring equipment, no other objects were present in the room.

Air speed measurements were recorded in three horizontal planes in the chamber: 0.1 m, 0.7 m and 1.3 m above the floor. In each plane, 12 measurements were recorded. These were located (see Fig. 2) below the centre of the fan ("centre"), below the perimeter of the fan ("*north, east, south, west*"), and on a radius at increasing distance from the centre of the fan. For instance, "*r 800*" means 800 mm away from the fan centre. Thus, in total, measurements were taken at 36 points (see Table 1).

Measurements were recorded simultaneously at these three heights in each of the 12 horizontal locations using three air speed probes (see Table 2 and Fig. 1). After the recording was completed in one location, the measurement equipment was moved to the next spot, but the new recording period always started a few minutes after having moved the equipment. The logging duration per location was 300 s and the logging period 1 s. It is important to point out that both types of probe used for measuring air speed are omnidirectional, which means that only air speed is recorded, rather than the three components of air velocity separately. Their measuring range is 0–10 m/s, in −20 to +70 °C range. Instantaneous measurements of room air temperature and relative humidity, room surface temperatures, and fan rotational speed have also been taken (see Table 3). In this study, the highest available fan