Building and Environment 141 (2018) 103–116

Contents lists available at ScienceDirect

Building and Environment

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

The effect of airspeed and wind direction on human's thermal conditions and air distribution around the body

Wonseok Oh[a,*], Shinsuke Kato[b]

[a] *Graduate School of Engineering, Department of Architecture, The University of Tokyo, 4-6-1 Komaba, Meguro-ku, Tokyo, 153-8505, Japan*
[b] *Institute of Industrial Science, The University of Tokyo, 4-6-1 Komaba, Meguro-ku, Tokyo, 153-8505, Japan*

ARTICLE INFO

Keywords:
Computational fluid dynamics (CFD)
Climate chamber
Wind tunnel
Equivalent temperature
Thermal manikin

ABSTRACT

The purpose of this research was to investigate the effects of airspeed and wind direction on a human's thermal conditions and of air distribution around the body. In addition, this study presents the equivalent temperature, which is the most effective index to evaluate the effect of air movement. The effect of convection and radiation on each part of the human body was estimated through numerical calculation, and the results were verified through a combination of thermal manikin experiments and radiation analysis. The experiments were conducted in a climate chamber and wind tunnel under conditions of natural convection and forced convection, respectively. An airspeed of 0.05 m/s was selected for the natural convection condition and an airspeed range of 0.2–2.0 m/s for forced convection conditions. The natural convective heat transfer coefficient in the stagnant environment was 3.27 $W\,m^{-2}K^{-1}$ in the sitting posture and 2.67 $W\,m^{-2}K^{-1}$ in the standing posture because the natural upward airflow caused by human body heat is more likely to diffuse to the surrounding environment in a sitting posture than in a standing posture. The equivalent temperature change of the whole body was greater in the standing posture than in the sitting posture when the airspeed was 0–0.2 m/s. The equivalent temperature change of the whole body was similar in both sitting and standing postures when the airspeed was greater than 0.2 m/s.

1. Introduction

Humans have a thermoregulation system that controls body temperature through physiological responses such as sweating, blood flow regulation, and shivering in response to a thermal environment. Heat gain occurs when the temperature of the external environment is higher than that of the skin temperature, and heat loss occurs when the temperature of the external environment is lower than that of the skin temperature. Humans maintain their core temperatures within a small range between 36 and 38 °C. Skin is the major organ that controls heat and moisture flow to and from the surrounding environment [1]. Humans feel comfortable at a thermal neutral state [2,3]. Within an environment with strong airflow or low air temperature, heat loss owing to convection increases. Understanding the effects of air movement on human's thermal conditions and its distribution around the body is very important to estimate the thermal sensation. As the human body itself is a heating element, it heats up the surrounding air by convection on its surface. The heated air assumes a natural ascending airflow owing to the buoyancy effect because of the temperature difference with the surrounding lower air temperature. The airflow flowing around the complex shape of a human body is difficult to predict when it is mixed with this natural ascending airflow. Air contaminants around humans are transferred along these flows and breathed into the lungs [4]. Moreover, the thermal sensation varies depending on the airflow characteristics. However, the airflow patterns in an indoor thermal environment are difficult to predict because of the diverse geometry and layout of furniture in a room. Moreover, various kinds of HVAC systems and occupants' behavior make airflow patterns more difficult to estimate. It is common to understand the effect of airflow on the human body through the analysis of a simplified airflow environment because numerous kinds of actual airflow patterns cannot be verified. Therefore, generally, the effect of the environment on each part of the human body is examined with uniform airflow conditions to predict the influence on the actual environment. Previous studies have been largely divided into computational fluid dynamics (CFD) simulation and experimental methods.

With regard to experimental methods, de Dear et al. [5] explained how the factors of airspeed and direction affect the radiative and convective heat transfer coefficients of each part of the human body. To separate the influence of radiation and convection, a method to

---

[*] Corresponding author.
E-mail addresses: oh-ws@iis.u-tokyo.ac.jp (W. Oh), kato@iis-u-tokyo.ac.jp (S. Kato).

https://doi.org/10.1016/j.buildenv.2018.05.052
Received 14 March 2018; Received in revised form 23 May 2018; Accepted 26 May 2018

determine the radiative heat transfer coefficient was proposed; the method involved changing the emissivity by wrapping aluminum foil on a thermal manikin. However, this is a time consuming and difficult task because repeated procedures are required to check the influence on each part of the body. A study on the heat transfer coefficient of the human body can help understand the degree of influence of airflow on a body. However, it is unclear how airflow distribution is formed around the human body by airspeed and wind direction and how it affects a human's thermal conditions. Oliveria et al. [6] studied the thermal effect of airspeed on the human body when a person was walking. With a thermal manikin experiment, the sensible heat loss and surface temperature owing to radiation and convection on the surface of the manikin were confirmed. The radiative heat transfer coefficient was calculated from the average manikin surface temperature and environment temperature obtained from the experiment using the analytical method in their research. By calculating the convective heat transfer coefficient with a combination of both methods, the effect of the airflow owing to the movement of a human while walking was verified. However, the study did not consider the direction of airflow and its formation around the human body. To experimentally examine how the variation of airspeed and its direction affect the airflow patterns around the human body, Licina et al. [7] used particle image visualization measurements complemented with a pseudo-color visualization technique. In their study, airflow distribution in a quiescent environment of less than 0.2 m/s was investigated by examining the influence of the changes in airspeed and airflow direction on the convective boundary layer around a human body. Although the study was deeply related to the airflow around the human body, the thermal effect of the human body owing to the airflow was not investigated. Furthermore, an airspeed condition of greater than 0.2 m/s was not considered.

CFD simulation is an effective way to overcome the limitation of the experimental method of measuring the effects of thermal environment on a human body and estimating the airflow patterns around the human body simultaneously. CFD shows reliable results compared with experiments for airflow patterns in a building. The first application of CFD to an indoor room was conducted by Nielsen [8] to predict airflow patterns. For the prediction of indoor air distribution, the RANS k–ε turbulence model has been used for many years and showed acceptable results in various research studies [9–11]. The standard k–ε turbulence model showed reasonable results for airflow prediction in indoor environments [12]. Moreover, compared to the standard k–ε model, the realizable k–ε model usually provides better results for swirling flows and flows involving separation [13–15]. These turbulence models based on high Reynolds numbers with wall functions showed good agreement over all with the airflow patterns in an indoor room. However, it is well known that the wall function approach results in inaccurate prediction of heat transfer in cases involving the anisotropy of near-wall turbulence, flow separation, and secondary motion in a flow field [16]. Therefore, in this study, the standard low Reynolds k–ε turbulence model [17] was used to predict the skin temperature of a thermal manikin and airflow patterns around the body more accurately.

With regard to the CFD simulation methods, Murakami et al. [18] examined the movement of contaminants in the airflow around a simplified human body standing in a stagnant environment. Gao et al. [19] confirmed the effect of airflow around a sitting human body model with the convective heat transfer coefficient. Murakami et al. [20] verified the thermal effects of the environment near a human body considering respiration. These studies focused on the effects of the natural ascending airflow generated by a warm human body in a stagnant environment. Yang et al. [21] and Ono et al. [22] investigated the effect of airflow on each part of the human body by using the convective heat transfer coefficient in a thermal manikin experiment and radiation analysis of a 3D manikin model. They focused on walking behavior and considered only frontal wind. The thermal effects on the human body based on the airflow direction and airflow patterns around the human body were not studied.

Many individual systems have been developed to improve the thermal sensation [23,24]. Among them, the ceiling fan has shown good results to improve human's thermal comfort in a warm humid climate. Research results showed occupants felt comfort with an airspeed of 1.2 m/s and a temperature of 30 °C [25]. In addition, a personal air supply system in commercial aircraft cabins and its control method (1.0–1.2 m/s) was proposed to enhance thermal comfort [26]. Furthermore, various kinds of personal comfort systems have shown good agreement with feeling comfort when the airspeed was high (0.8–1.6 m/s and higher than 1.6 m/s) at high temperatures [27]. We examined the airspeed conditions in the range of 0.05 (stagnant condition) to 2.0 m/s to verify how the various individual systems thermally affect each part of the human body.

In addition, we considered horizontal wind blowing from the front, back, and cross directions considering the acceptability of personal comfort systems [24] such as a desk fan [23,28], a chair with a fan for personal comfort [29,30], the air vent of vehicles, and ventilation by window opening.

2. Method

2.1. Numerical method

2.1.1. Natural convection condition

In the CFD analysis, STAR-CCM + software based on the finite volume method was utilized. Table 1 presents details of the simulation method in a stagnant environment. The Navier–Stokes equations were solved using the standard low Reynolds number k-ε turbulence model [31]. The surface-to-surface model was used for radiation analysis. The pressure-velocity coupling for the airflow solution was conducted with the SIMPLE algorithm [32]. The convection and diffusion terms were integrated using the QUICK difference scheme [33]. Prism layers were utilized so that the entire surface of the manikin was $y+ < 1$.

Simulation conditions of the stagnant environment were set by referring to the laboratory conditions for a fair comparison with the results of full-scale measurements. The manikin model was located at the center of an indoor space of 3.0 m × 3.5 m × 2.5 m (Fig. 1). Two manikin postures were considered: sitting and standing. The airflow was controlled at 0.05 m/s and 24 °C and introduced through the total area of the room floor (section ABCD). The exhaust condition was set to be exhaust through section EFGH.

The grid systems are shown in Fig. 2. To calculate the boundary layer around the manikin model, 30 prism layers were located at the surface of the manikin body with a thickness of 0.02 m and growth rate of 1.13 in each cell layer. The grid system of the sitting posture simulation was performed in a similar manner.

Table 2 presents the boundary conditions of the stagnant environment. The turbulence intensity of the inlet air was set as 10% and the turbulence length scale as 0.1 m. The exhaust was set at the pressure condition (pressure = 0). Since the surface of the room was not of interest in this study, it was set as an adiabatic wall to stabilize to the same temperature condition as the indoor air. Although the influence of

Table 1
Numerical methods for manikin simulation.

Commercial code	STAR-CCM + Ver. 12.02
Turbulence model	Standard low Reynolds k–ε turbulence model
Numerical methods	Velocity-pressure correction: SIMPLE algorithm
	Convection scheme: second-order upwind
Buoyancy	Boussinesq approximation
Radiation calculation	Surface-to-surface model
Grid system	Shape: tetrahedral mesh, prism layer
	Manikin body (30 layers, $y+ < 1$)
	Number of cells: approximately 20 million