Energy and Buildings 104 (2015) 199–207

Contents lists available at ScienceDirect

Energy and Buildings

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

Numerical simulation of airflow and temperature fields around an occupant in indoor environment

Javad Taghinia*, Md Mizanur Rahman, Timo Siikonen

Aalto University, School of Engineering, Department of Applied Mechanics, Finland

A R T I C L E   I N F O

Article history:
Received 10 February 2015
Received in revised form 14 June 2015
Accepted 27 June 2015
Available online 11 July 2015

Keywords:
Displacement ventilation
CFD
LES
One-equation model
Indoor environment

A B S T R A C T

An accurate prediction of flow and thermal fields around an human body provides valuable information in designing an efficient ventilation system. This study tries to address this issue by investigating the flow structure around an human body subjected to a displacement ventilation system through the computational fluid dynamics (CFD). For this purpose, both the large eddy simulation (LES) and hybrid LES–RANS methods are employed. The RAST one-equation model (OEM) and the dynamic Smagorinsky model (DSM) are utilized as a sub-grid scale (SGS) modeling for the LES while for the hybrid LES–RANS method, the SST–SAS version is applied. Predicted results are compared with available experimental measurements in the literature. Comparisons show that the OEM has a better performance in reproducing the correct level of velocity and temperature fields around the body as well as in other locations of the room while the SST–SAS model fails to predict these quantities accurately, especially around the occupant's body. Above all, the OEM provides a good compromise between accuracy and robustness in predicting the airflow and temperature fields in an indoor environment.

1. Introduction

Air distribution in a room depends on many influential parameters, such as the air supply condition, number of occupants, room geometry, method of ventilation and number of diffusers [1]. Each of these factors affects the air quality in a dramatic way, causing an impact on the indoor air quality (IAQ) [2]. In the design of any energy efficient ventilation systems, the interaction of air and occupant plays a vital role because in most scenarios, the final goal is to achieve the thermal comfort of occupants in enclosed spaces [3]. Therefore, in order to determine the best ventilation process, the effect of occupants on the flow behavior should be accounted for in the analysis. Apart from the effect of occupants' body on distorting the air movement, the human skin exchanges heat with the surrounding air, creating a plume of air around the body. This plume (mainly due to the buoyancy) can affect the velocity and temperature distributions around the occupant that contribute to the mixing process in a room [2].

Experimental procedures provide an accurate understanding of the air behavior inside a room; however they are expensive and time consuming. On the other hand, the computational fluid dynamics (CFD) is much less expensive (depending on the applied method and geometry) and can predict some detail information on the structure of the airflow which is not easily achievable in experimental measurements. It is worth mentioning that CFD results can be misleading if a proper attention is not paid to a few significant parameters in the modeling, such as initial values, boundary conditions and computational grid qualities [4].

There are several studies in the literature concerning the analysis of an indoor air behavior in the building and residential spaces [5–9]. One of the first efforts in applying the CFD, related to the interaction of occupant and indoor air was made by Murakami et al. [10]. They applied different settings and conditions to investigate the flow field around an unclothed manikin with a displacement ventilation (DV) scenario. Nilsson and Holmer [1] applied the RNG k–ϵ model to simulate the flow around a seated manikin, subjected to a displacement ventilation system. They used their simulated data to formulate a thermal comfort index. Murakami et al. [11] studied the flow around a standing person by a low-Reynolds number k–ϵ model with thermal radiation effects. As a benchmark test, Sideroff and Dang [12] applied the standard k–ϵ and v^2–f turbulence models to analyze the flow topology around an occupant, exposed to a displacement ventilation system. Their study showed that the v^2–f model could produce better predictions compared to the k–ϵ model. In majority of these numerical studies, the simplified geometry of an human body with rectangular shapes was

---

* Corresponding author.
E-mail address: javad.taghiniaseyedjalali@aalto.fi (J. Taghinia).

http://dx.doi.org/10.1016/j.enbuild.2015.06.085
0378-7788/

Nomenclature

C_μ	eddy-viscosity coefficient
$\overline{C}_s$	Smagorinsky coefficient
G	filter function
g	gravitational acceleration
k	total turbulent kinetic energy
L_{ij}	Leonard stress
Pr	molecular Prandtl number
Pr_{sgs}	sub-grid scale Prandtl number
Re	Reynolds number
$\overline{S}_{ij}$	mean strain-rate tensor
T	temperature
$\overline{u}_i$	grid-filter velocities
$\tilde{\overline{u}}_i$	test-filter velocities
$\overline{u}_\tau$	friction velocity
$\overline{W}_{ij}$	mean vorticity tensor
y^+	dimensionless wall distance ($\overline{u}_\tau y/\nu$)
β	thermal expansion coefficient
$\delta_{i,j}$	Kronecker's delta function
Δt	time step
$\overline{\Delta}$	grid-filter width
$\tilde{\Delta}$	test-filter width
ν, ν_T	laminar and turbulent viscosities
$\overline{\theta}_i$	grid-filter temperature
$\tilde{\overline{\theta}}_i$	test-filter temperature
ρ	density
$\tau_{i,j}$	sub-grid scale stress tensor
LES	large eddy simulation
RANS	Reynolds averaged Navier–Stokes
RAST	Rahman–Agarwal–Siikonen–Taghinia
DSM	Dynamic Smagorinsky Model
SGS	sub-grid scale

Subscript

i, j	variable numbers
in	inlet condition
out	outlet condition

considered. This type of geometry was used in the work of Villi and De Carli [2] in which the real human body was replaced with four rectangular blocks. In their study, they applied a fire dynamics simulator (FDS) solver, utilizing a large eddy simulation (LES) model. They obtained satisfactory results for the temperature distribution in various locations, however, the predicted velocity profiles especially close to the occupant were not acceptable when compared with measurements. Deevy [13] investigated the effect of various human-body geometries on the flow field in a room. He considered three levels of a geometry, ranging from a simple cylinder to a complex geometry for modeling the human body. His results showed that all levels of the geometry produced similar results away from the body while for the simplest geometry, the predictions were not good at critical locations such as above the head. Deevy et al. [14] applied the SST–unsteady RANS (URANS) and the detached eddy simulation (DES) to investigate the flow and thermal fields around a realistic human body. The URANS results showed excessive velocity magnitudes close to the top and bottom walls. Comparisons with the experimental data proved that the SST–URANS is unable to accurately predict the temperature profiles at different locations in the room while the DES produced more reasonable results. Sideroff and Dang [15] conducted a number of simulations with different grid resolutions based on guidelines of a benchmark displacement ventilation case of Nilsson and Holmer [1]. They applied an unstructured mesh for all cases for a realistic human body using the v^2–f model and the dynamic Smagorinsky model (DSM). They reported that obtaining grid independent solutions while keeping a reasonable cell-count and using only a tetrahedral mesh were extremely difficult. They concluded that the DSM results showed no advantages compared to those of v^2–f. Srebric et al. [4] simulated a simplified human body (consisting of cuboids) applying the standard and RNG k–ϵ models with 6.5×10^5 control volumes. They noticed that the simplified geometry was satisfactory in reproducing some basic parameters according to the measurements. The flow around a modeled seated occupant subjected to a displacement ventilation system was investigated by Salmanzadeh et al. [16] using the standard k–ϵ model with a Lagrangian particle transport model. Their study showed that the buoyancy driven thermal plume close to the body strongly affected the airflow pattern. Ge et al. [17] conducted a simulation with the RNG k–ϵ model for the flow field around a standing human for various incoming velocities. They found satisfactorily results for main parameters such as the velocity distribution and concluded that the buoyancy effect is stronger in cases with an incoming velocity less than 0.2 m/s.

Considering the above-mentioned literature review, it is evident that the applied RANS models are unable to capture a correct behavior of airflow in the room, particularly close to the occupant. However, there are a few studies concerning the LES; they are mainly based on the dynamic Smagorinsky sub-grid scale (SGS) model. Apart from this fact, most of the works done on this subject, considered a simplified model of an human body which may cause unrealistic results in terms of flow and thermal behaviors. This paper aims at assessing the capability of three models in simulating the airflow and thermal fields in a room with an occupant subjected to a displacement ventilation system, namely, the RAST (Rahman–Agarwal–Siikonen–Taghinia) one-equation model (OEM) [18], dynamic Smagorinsky model (DSM) of Germano [19] and SST–SAS model. The former two models are categorized as the SGS modeling approach in an LES and the latter one belongs to an hybrid LES–RANS method. As there are no reported results on the in-depth performance of these models (especially OEM and SST–SAS), it seems essential to provide a benchmark study on this subject. Another advantage of this study is the use of a real human-body geometry which provides a more realistic computational flow condition. The predicted results are compared with the existing experimental data in terms of velocity and temperature distributions.

2. Governing equations

2.1. RAST one-equation SGS model

A spatial filter is used in an LES to separate the large scales from the small scales that are to be modeled. By applying a spatial filter to incompressible Navier–Stokes equations and using the commutation characteristics, the LES equations yield:

$$\frac{\partial \overline{u}_j}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i \overline{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \partial \frac{\overline{u}_i}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} + g_j \beta(\overline{\theta} - \theta_\infty)\delta_{ij} \tag{2}$$

where the subscript (∞) refers to the reference condition, g_j denotes the gravitational acceleration and β is the volumetric thermal expansion coefficient (obtained from appropriate property tables). The *overbar* notation denotes the application of a top-hat filter and ν is the kinematic viscosity. Since in the LES formulation the larger length scales are resolved, it appears that the turbulent SGS stresses