Building and Environment 144 (2018) 259–267

Contents lists available at ScienceDirect

Building and Environment

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

Evaluation of models and methods to simulate thermal radiation in indoor spaces

Xiangdong Li[a], Yihuan Yan[a], Jiyuan Tu[a,b,*]

[a] School of Engineering, RMIT University, PO Box 71, Bundoora, VIC, 3083, Australia
[b] Key Laboratory of Ministry of Education for Advanced Reactor Engineering and Safety, Institute of Nuclear and New Energy Technology, Tsinghua University, PO Box 1021, Beijing, 100086, China

ARTICLE INFO

Keywords:
Radiative heat transfer
Theoretical models
Indoor environment
CFD
Computational cost

ABSTRACT

The theoretical models of Surface-to-Surface (S2S) thermal radiation, including the Monte Carlo model, Discrete Transfer model, Modest model and Heat-Flux-Split approach, are evaluated in terms of the predictive accuracy and CFD computational cost when simulating indoor thermal flows. It is demonstrated that the inclusion of thermal radiation in the CFD model is vital as the air temperature in the lower levels can be underpredicted while the heater surface temperature can be significantly overpredicted if the radiative effects are ignored. In addition, the predicted temperature distribution on the heat-receiving solid surfaces is highly sensitive to the selected radiation model. The comparisons demonstrate that the Monte Carlo model and Discrete Transfer model have comparable predictive capabilities while the latter requires less CPU time and is more computationally efficient. The appropriate number of the representative photons and rays are also recommended for the Monte Carlo model and Discrete Transfer model, respectively.

1. Introduction

Thermal radiation is the spontaneous emission of energy from any matter that has a temperature greater than the absolute zero. Due to the 4th-order dependence of radiative heat flux on temperature, the radiative mechanism can sometimes transfer a significant portion of heat under room-temperature conditions. For instance, the percentage of heat dissipation through thermal radiation from human bodies ranges from 30% to 70% [1,2] in office rooms. For industrial buildings in which furnaces, boilers and gas turbine combustors exist, thermal radiation accounts for 80%–90% of the total heat exchange [3]. It is reasonable to expect that thermal radiation may play a critical role in shaping the indoor environment quality (IEQ) in terms of airflow pattern, thermal comfort, contaminant dispersion and air quality.

However, when it comes to the studies of indoor airflow using computational fluid dynamics (CFD), the effects of interior thermal radiation have been commonly ignored [4,5]. Some studies stated that it is safe to ignore thermal radiation when the surface area of heat sources is small compared to that of heat-gaining surfaces [2], or when the ventilation rate is high such as that in operating rooms where the air change rate could be as high as 40 h^{-1} [6]. Ignoring the radiative effects, although may facilitate model validation through small-scale experiments using liquids [7], can result in considerable predictive inaccuracy, particularly in low-momentum spaces such as those with natural or displacement ventilation schemes [3,8].

Li et al. [9] simulated the buoyance-driven flow in a small rectangular cavity. The results showed that when radiation was added to the computational model, a strong flow was induced near the floor and the thermal stratification in the cavity was less pronounced, resulting in a warmer bulk region. Using a cubic cavity model, Kogawa et al. [10] demonstrated that the radiative heat transfer between solid surfaces was dominant in the generation of shear stress by turbulent flows. A recent study of Menchaca-Brandan et al. [7] revealed that when radiative heat transfer was ignored, the air temperature surrounding human occupants was underpredicted by 2–4 °C while the surface temperature of heat sources was significantly overpredicted by up to 17 °C. In addition, Tam et al. [11] studied the indoor conditions of a rectangular single-story building. They found that the radiative heat transfer not just had a significant effect on the predicted contaminant concentration fields, but also strongly affected the energy simulation results.

To numerically account for the radiative effects on indoor thermal flow fields, various approaches and models have been developed. As an approximation, Srebric et al. [2] recommended a 30:70 convection to radiation split of the human metabolic heat in room models. They considered the radiative effects in the CFD model through equally

---

[*] Corresponding author. School of Engineering, RMIT University, PO Box 71, Bundoora, VIC, 3083, Australia.
E-mail address: jiyuan.tu@rmit.edu.au (J. Tu).

https://doi.org/10.1016/j.buildenv.2018.08.033
Received 26 June 2018; Received in revised form 31 July 2018; Accepted 19 August 2018

applying the 70% radiative heat flux on the solid surfaces. This approach was also utilized by Yan et al. [4,12] to model the human thermal plume where a reasonable convective heat flux is needed. However, the actual ratio of convective to radiative heat fluxes depends strongly on the room conditions and can be very dynamic and unevenly distributed. A fixed value is apparently not a good estimation of the physics.

At the meantime, radiative heat transfer is also extensively modelled using the Radiation Transport Equation (RTE) [8,13]. The RTE is regarded as a mechanistic approach as it is able take into account the room geometry and spectrum features of the participating media. However, the RTE-based models are computationally very expensive as the solution of the RTE is iterative and dependent on the three spatial coordinates, 2 location directions and the frequency and wavelength. Yousaf et al. [14] found that the CPU time required to solve the RTE increased exponentially with the number of discrete rays, so that the computational cost was very high in order to achieve an acceptable thermal comfort assessment. On the top of that, the poor numerical stability induced by the coupling of the 4th-power radiation and 1st-power convection models further impedes integrated analyses [6]. Although new techniques such as programmable graphics hardware [15] have in recent years been utilized to enhance the computing capability, the high computational cost remains a major challenge when modelling indoor airflows with radiative heat transfer.

In fact, the concentration of radiation-participating molecules such as H_2O and CO_2 is very low in common indoor air [16], so that the air is optically very transparent in the infrared spectrum. In this case, any emission, absorption or scattering of radiative energy by the air can be safely ignored and the thermal radiation can only affect the airflow field through heating or cooling the solid surfaces. Therefore, the effects of thermal radiation can be approximately modelled via properly formulated boundary conditions at the fluid-solid interfaces using the so-called Surface-to-Surface (S2S) approach [17], which could be much more cost-efficient than solving the RTE. The most popular S2S models [18] include are the Monte Carlo (MC) model and Discrete Transfer (DT) model, while some other models and approaches [2,18] have also been developed to estimate the radiative heat exchange in indoor spaces.

However, to the best of our knowledge, these models and approaches have never been compared against each in terms of predictive accuracy and computational cost, which has sometimes made it hard for HVAC engineers to select an appropriate model when designing built environments. Therefore, this study presents a numerical evaluation of these models in terms of computational cost and accuracy.

2. Models of surface radiation in indoor environments

In a S2S radiative heat transfer model, all the solid surfaces are assumed to be gray and diffuse, meaning that their emissivity and absorptivity are independent of the wavelength and the reflectivity is independent of the outgoing or incoming directions. For an opaque gray-diffuse surface, its emissivity equals to the absorptivity ($a = \varepsilon$). The energy incident on the surface is partially absorbed and reflected ($a + \rho = 1$). Therefore, for the kth ($k = 1$ - N) adiabatic surface in the computational domain, the energy flux leaving it is composed of the directly emitted and reflected energy.

$$q_{out,k} = \varepsilon_k \sigma T_k^4 + (1 - \varepsilon_k) q_{in,k} \tag{1}$$

where, $q_{out,k}$ is the heat flux leaving the surface, σ is the Stefan-Boltzmann constant and T_k is the local surface temperature. $q_{in,k}$ is the energy flux incident on the surface from the surroundings.

In the CFD computations, the radiative heat transfer was jointly solved with the airflow field. To do this, the incident heat flux was incorporated in the CFD model as a boundary condition of the energy equation. The local energy balance of the kth ($k = 1$ - N) grid element on the solid surface was expressed by

$$q_{ext} + q_{in,k} - (1 - \varepsilon_k) q_{in,k} - h_k (T_k - T_a) - \varepsilon_k \sigma T_k^4 = 0 \tag{2}$$

where, q_{ext} is the flux of externally applied heat, h_k is the convective heat transfer coefficient and T_a is the air temperature. The items on the left-hand side of Eq. (2) represents the externally applied, incident, reflected, convective and directly emitted heat flux components, respectively.

2.1. The Monte Carlo (MC) model

The incident energy can be modelled using a view factor, which is defined as the fraction of energy leaving one surface that is incident on another surface. For the kth surface in the domain, the incident energy can be expressed in terms of the energy flux leaving all the other surfaces

$$A_k q_{in,k} = \sum_{j=1}^{N} A_j q_{out,j} F_{jk} \tag{3}$$

where, A_k and A_j are areas of surface k and j, respectively. F_{jk} is the view factor from surface j to surface k. Due to the view factor reciprocity relationship [19], we have

$$A_k F_{jk} = A_j F_{kj} \tag{4}$$

And

$$q_{in,k} = \sum_{j=1}^{N} q_{out,j} F_{kj} \tag{5}$$

The view factor was calculated in this study using the Monte Carlo method [13]. The basic idea of the Monte Carlo model is that the solid surface is randomly emitting photons towards the space. Statistically, the probability of the photons hitting another surface represents the fraction of energy leaving one surface that is incident on another surface [20], namely the view factor. As shown in Fig. 1, surface element dA_j is emitting photons into a semi-infinite space. When the surface is diffuse, the total radiation intensity is independent of the cone angle β ($\beta = 0 \sim \pi/2$) and the azimuth angle φ ($\varphi = 0 \sim 2\pi$). Therefore, the probability of a photon travelling in a definite β direction is defined by Ref. [21].

$$p(\beta) = \frac{\cos\beta \sin\beta}{\int_0^{2\pi} \cos\beta \sin\beta d\beta} = 2\cos\beta \sin\beta \tag{6}$$

The equation describes the spatial distribution of the radiative energy from and upon a diffuse surface. In this study, a randomly generated number R_β ($R_\beta = 0$–1) is used to assure the demographic

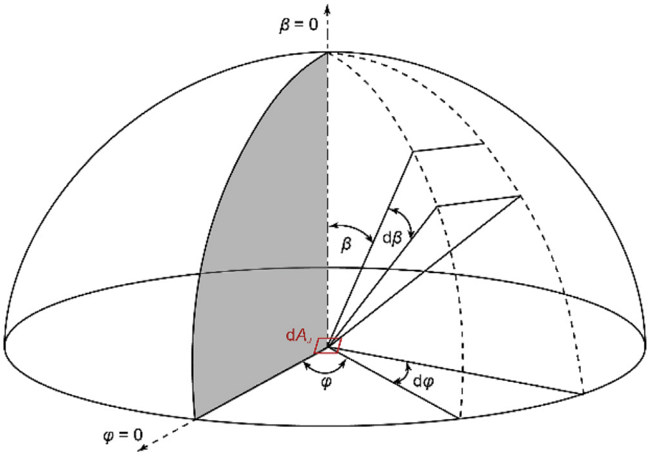

Fig. 1. Probability of radiation travelling in a definite direction.