How safe is it to neglect thermal radiation in indoor environment modeling with high ventilation rates?

Sasan Sadrizadeh* **and Sture Holmberg**

Division of Fluid and Climate Technology, School of Architecture and the Built Environment, KTH Royal Institute of Technology, Stockholm, Sweden

*Corresponding email: ssad@kth.se

ABSTRACT

Typical heat sources in indoor environments include humans, electrical devices, and computers. The number of such sources in operating room environments is even higher due to the presence of surgical staff members and medical equipment. The exchange of thermal energy between indoor surfaces and air is usually modelled by considering contributions from both radiation and convection. Complete heat transfer simulations in indoor environments are normally difficult since radiation models have a tendency to generate numerical instability and, hence, problems with convergent solutions. In the past, thermal radiation influence on indoor airflow movement and contaminant distribution was rarely addressed. This study therefore focused on evaluating the influence of radiative heat transfer in an operating room with 40 air changes per hour.

For the purpose of the current study, an identical case with and without considering the radiative part of heat transfer was simulated and compared. In both simulated cases, the numerical results have shown negligible changes in terms of indoor temperature and air velocity, and thus contaminant distribution. It was also observed that temperature differences between heat-emitting surfaces were negligible. It is therefore reasonable to conclude that it is safe to disregard the radiative part of heat transfer in indoor air simulations with high ventilation rates and heat transfer can here be modelled as pure convection.

KEYWORDS: heat transfer, radiation modeling, operating room, contaminant (particle) distribution, computational fluid dynamics (CFD), numerical simulation

1. INTRODUCTION

Indoor airflow simulations can predict contaminant dispersion and provide valuable information regarding indoor air quality, especially in sensitive environments such as operating rooms (ORs). With recent advances in computer capability and speed, computational fluid dynamics (CFD) has become a powerful alternative for predicting airflows in enclosed environments. Simulation accuracy greatly depends on the proper setting of boundary conditions and numerical simulation parameters.

The precise setting of thermal boundary conditions is required for correct prediction of indoor environment temperature distribution. This may result in precise airflow calculation and thus airborne particle concentration. A local indoor temperature that is miscalculated by 2 °C can result in determining an incorrect airflow pattern, even opposite direction compared to the real airflow patterns (Yuan & Srebric, 2004). This may generate substantial errors in determining the distribution of indoor contaminants, which usually follow airflow streamlines closely (Sadrizadeh & Holmberg, 2014a; Sadrizadeh, Tammelin, Ekolind, & Holmberg, 2014). Typical CFD simulations were usually considered the sensible heat loads as 100 % convective (Sadrizadeh, Holmberg, & Tammelin, 2014). This was a simplified treatment since a portion of the sensible load should be radiative in nature.

Complete heat transfer simulations in indoor environments are difficult since radiation modelling has a tendency to generate numerical instability and thus problems with convergent solutions. Thus, thermal radiation influence on indoor airflow simulations was seldom addressed.

This study is therefore focused on evaluating the influence of radiative part of heat transfer on temperature and airflow velocity in an operating room with 40 air changes per hour (ACH).

2. METHODOLOGIES

A single-zone standard OR, which was adapted from the authors' previous work (Sadrizadeh & Holmberg, 2014b; Sadrizadeh, Tammelin, Nielsen, & Holmberg, 2014), was chosen as the physical model for this study. The OR measured 8.5 m × 7.7 m × 3.2 m (H), with the physical configuration shown in Fig. 1.

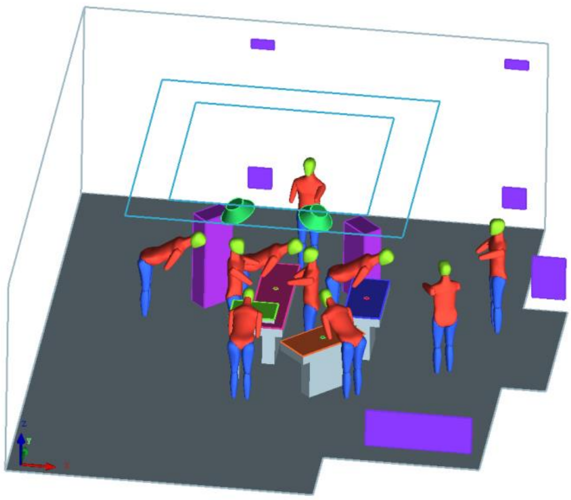

Fig. 1: Isometric view of operating room

The supply air was directed to the OR, with a total volume flow rate of 2,000 L/s, giving a design exchange rate of 40 h^{-1}. Supply air temperature and turbulence intensity were 20 °C and 5 percent respectively. Exhaust air was expelled out through six grilles placed on three parallel vertical walls. An operating table, two instrument tables, and one mayo stand table were considered within the surgical zone. Two pieces of medical equipment, each with thermal load of 255 W, were also included. Two medical lamps were fixed above surgical area; each emitted a heat load of 320 W. Ten staff members were considered in both "bend" and "straighten up" postures. Any of the simulated staff members with 1.6 m^2 total surface area emits a thermal heat load of 195 W.

The mean airflow field was solved using the realizable k-ε turbulence model. This model is widely used for indoor environments due to its relative simplicity and robustness for indoor airflow simulations. The governing equations can be written in the general format as follows:

$$\frac{\partial}{\partial t}(\rho\varphi) + \nabla.\left(\rho\varphi\vec{V} - \Gamma_{\varphi}\nabla\varphi\right) = S_{\varphi} \qquad \text{Eq. 1}$$

where ρ is the air density, φ represents each of the three velocity components u, v, w and $\vec{V}$ is the velocity vector. S_{φ} is the source term and Γ_{φ} is the effective diffusion coefficient for each dependent variable.

The above-mentioned equations are discretized into algebraic equations by the finite volume method and solved by ANSYS Fluent 15.0. The discretized method for convection terms is a second-order upwind scheme. The SIMPLE algorithm was adapted to couple pressure and velocity. Enhanced wall treatment was employed to treat the turbulent airflow properties in the near-wall regions. The numerical models were previously validated against experimental data (Sadrizadeh, Holmberg, et al., 2014; Sadrizadeh, Tammelin, Ekolind, et al., 2014) and are not repeated here.

In the current study, the sensible heat loads of an identical case were considered once as "100 % convective" and once as "combination of radiative and convective". For this purpose, radiative heat transfer was simulated using discrete ordinates (DO) radiation model theory, which solves the radiation transfer equation for a finite number of discrete solid angles (in this study 32). Here, all rigid surfaces were treated as sources of opaque gray (emissivity of 0.95) radiation.

3. RESULTS AND DISCUSSIONS

In general, particle distribution is highly dependent on Stokes number (Stk). Stk is defined as the ratio of the characteristic time of a droplet to a characteristic time of the airflow or of an obstacle. Particles with Stk < 0.1 will follow airflow streamlines closely (Gao & Niu, 2007). The authors have previously shown that the Stocks number of common particles with the surgical area have the stocks number of less than 10^{-3} (Sadrizadeh & Holmberg, 2014a; Sadrizadeh, Tammelin, Ekolind, et al., 2014); therefore, the contaminants will follow the airflow streamlines.

Fig. 2 shows the location of eight lines across the OR, which the airflow temperature and velocities exported.

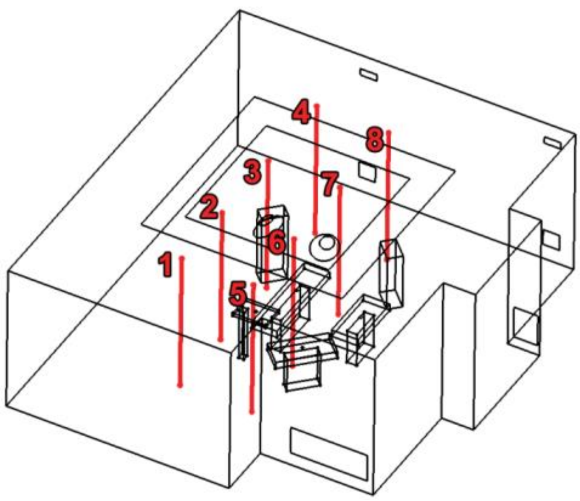

Fig. 2: Eight vertical lines representing the location of exported temperature and velocity data

Fig. 3 shows the ratio of the temperature and velocity magnitude, with (T_r, V_r) and without (T, V) considering the radiative part of heat transfer.

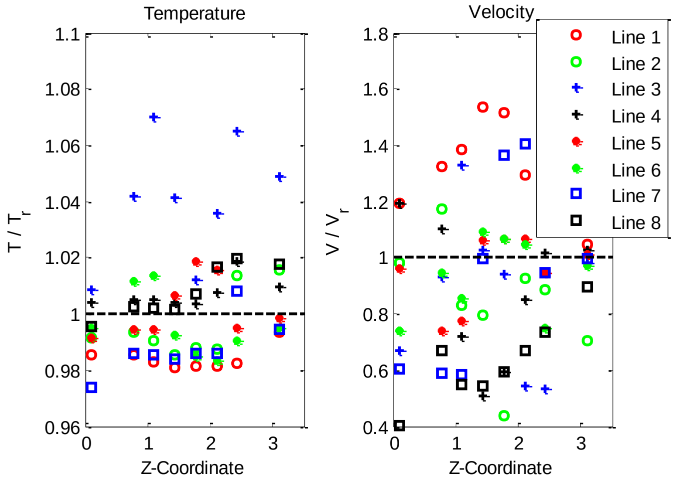

Fig. 3: Ratio of temperatures, one with considering the radiative heat transfer (T_r) and one without (T) [left]; and ratio of velocities with (V_r) and without (V) considering the radiative heat transfer [right]

It is clear that the temperature difference between the results obtained with and without considered the radiative part of heat transfer is negligible. This difference is much higher in case of the velocity ratio (V/V_r). This might be due to the velocities being less than 0.4 m/s, as small differences may result in quite high ratios. In operating room environments, since the air exchange rate is usually very high compared to the normal indoor air situations, the convective part of heat transfer become highly dominant. Therefore, most of the heat transfer will be handled by convection. Radiative part heat transfer thus has a negligible effect and can be disregarded.

4. CONCLUSIONS

In order to access the effect of thermal radiation in indoor airflow simulations with high ventilation rates, an identical operating room was simulated. The sensible heat loads were either taken as 100 percent convective or a combination of radiative and convective. The discrete radiation model coordinates handle the thermal radiation effect. In both cases, the numerical results indicate negligible changes in indoor air velocity and temperature distribution and thus on particle concentration. However, simulation of velocity and contaminants may require further assessment in establishing the influence of simplifications.

5. REFERENCES

Gao, N. P., & Niu, J. L. (2007). Modeling particle dispersion and deposition in indoor environments. Atmospheric Environment, 41(18), 3862–3876.

Sadrizadeh, S., & Holmberg, S. (2014a). Effect of a portable ultra-clean exponential airflow unit on the particle distribution in an operating room. Particuology, DOI: 10.1016/j.partic.2014.06.002, [In Press].

Sadrizadeh, S., & Holmberg, S. (2014b). Surgical clothing systems in laminar airflow operating room: a numerical assessment. Journal of Infection and Public Health, 7(6), 508–516.

Sadrizadeh, S., Holmberg, S., & Tammelin, A. (2014). A comparison of vertical and horizontal laminar ventilation systems in an operating room: A numerical study. Building and Environment, 82, 517–525.

Sadrizadeh, S., Tammelin, A., Ekolind, P., & Holmberg, S. (2014). Influence of staff number and internal constellation on surgical site infection in an operating room. Particuology, 13, 42–51.

Sadrizadeh, S., Tammelin, A., Nielsen, P. V, & Holmberg, S. (2014). Does a mobile laminar airflow screen reduce bacterial contamination in the operating room? a numerical study using computational fluid dynamics technique. Patient Safety in Surgery, 8(1), 27.

Yuan, J., & Srebric, J. (2004). Transient prediction of contaminant distribution by introducing energy load calculations into multi-zone modeling. In CIB World Building Congress 2004 (Vol. 148, pp. 2–7). Toronto, Canada.