Available online at www.sciencedirect.com

ScienceDirect

Building and Environment 44 (2009) 44–55

BUILDING AND ENVIRONMENT

www.elsevier.com/locate/buildenv

Transport of exhaled particulate matter in airborne infection isolation rooms

Jennifer Richmond-Bryant*

Environmental and Occupational Health Sciences, Hunter College, City University of New York, 425 East 25th Street, New York City, NY 10010, USA

Received 30 July 2007; received in revised form 22 January 2008; accepted 23 January 2008

Abstract

The goal of this research was to examine the characteristics of the spatial velocity and concentration profiles which might result in health care workers' exposure to a pathogenic agent in an airborne infection isolation room (AIIR). Computational fluid dynamics simulations were performed for this purpose. This investigation expanded on the work of Huang and Tsao [The influence of air motion on bacteria removal in negative pressure isolation rooms. HVAC & R Research 2005; 11: 563–85], who studied how ventilation conditions impact dispersion of pathogenic nuclei in an AIIR by investigating the airflow conditions impacting dispersion of infectious agents in the AIIR. The work included a careful quality assurance study of the computed airflow, and final simulations were performed on a fine tetrahedral mesh with approximately 1.3×10^6 cells. The 1 μm diameter particles were released from a 0.001225 m^2 area representing the nose and mouth. Two cases were investigated during the current study: continuous exhalation of pathogen-laden air from the patient and expulsion of pathogenic particles by a single cough or sneeze. Slow decay of particle concentration in the AIIR during the single cough/sneeze simulation and tendency for particle accumulation near the AIIR walls observed in the continuous breathing simulation suggest that unintended exposures are possible despite the ventilation system. Based on these findings, it is recommended that extra care be taken to assure proper functionality of personal protective equipment used in an AIIR.

Keywords: Airborne infection isolation room; Aerosol transport; Computational fluid dynamics; Indoor air quality; Particle transport

1. Introduction

A number of administrative and engineering controls are commonly employed to protect health care workers and other hospital patients from exposure to highly infectious airborne pathogens [1,2]. Airborne infection isolation rooms (AIIRs) are designed to protect facility occupants outside the room. A separate ventilation system is used for the AIIR to prevent mixing of potentially contaminated air with fresh intake air for the ventilation system serving the majority of the hospital. The AIIR infiltration and exhaust air supplies maintain negative pressure within the room to prevent the pathogens from entering other parts of the hospital and to minimize concentrations of the pathogen inside the AIIR.

Within the AIIR, administrative controls are typically used for worker protection. These include use of personal protective equipment (PPE) and thorough hand disinfection. Type N_{95} respirators are routinely used to prevent health care workers' exposure to airborne nuclei of infectious bacterial or viral pathogens [3]. Potential for exposure exists when respirators do not fit the employees properly. Face seal leakage can occur under a variety of conditions, such as when fit testing is inadequate, the healthcare worker has gained or lost weight since the most recent fit test, or the healthcare worker is not cleanly shaven when visiting infectious disease patients [4]. Moreover, Yen et al. [1] maintain that, while important, the standard protocols for preventing infectious disease may not be sufficient. Ofner et al. [5] found that health care workers were infected with severe acute respiratory syndrome (SARS) when wearing PPE that was believed to be sufficient. Hence, administrative controls may not be adequate for health care workers entering AIIRs, even if

---

*Tel.: +1 212 481 7580; fax: +1 212 481 5260.
E-mail address: jrichmon@hunter.cuny.edu

0360-1323/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.buildenv.2008.01.009

the AIIR protects occupants of the remainder of the hospital.

Given the concern for lack of protection within an AIIR, it is important to develop an understanding of air and contaminant transport in the room. A comprehensive review of studies of airborne transmission of infectious agents has demonstrated conclusive evidence that air movement can contribute to transmission and subsequent outbreak of infectious disease [6]. Laboratory and computational research has shown that the well-mixed assumption is not necessarily valid immediately after an aerosol is released [7,8]. Instead, non-dimensional aerosol residence time has been shown to be a function of turbulence intensity, which in turn is a function of positioning of the ventilation inlets and outlets and furnishings. In support of that point, transmission of exhaled air from one body to another and resulting airborne concentration have been shown for displacement ventilation and hospital rooms to depend on the position of the source, turbulence caused by movement of mannequins, and turbulence caused by the ventilated air interacting with surfaces and heat plumes in a room [9,10]. Other computational studies have demonstrated a relationship between the aerosol residence time and convective transport [11]. Knowledge of the airflow patterns may thus elucidate concentration gradients. Graded distribution of the pathogen concentration with respect to the patient's breathing zone may facilitate creation of a spatial probability exposure distribution within the room. This, in turn, may guide determination of safe behavior practices within an AIIR.

Given the sensitivity of taking and handling biological aerosol samples in an AIIR, computational fluid dynamics (CFD) analysis techniques are especially appropriate for the problem of highly infectious pathogen particulate transport. CFD can provide a highly resolved estimate of air and species transport within the region of interest in lieu of obtaining field samples, because AIIRs may be difficult to access and samples may be dangerous to obtain and analyze. CFD has been employed to compare ventilation designs in hospital rooms [10] and AIIRs [12–14], study the dispersion of tuberculosis nuclei [15], and assess the effect of human movement on pathogen dispersion in an AIIR [16].

The goal of this analysis was to examine the characteristics of the spatial velocity and concentration profiles, which might lead the health care worker to be exposed to a pathogenic agent in an AIIR. The study expanded on the work of Huang and Tsao [12] which studies the removal of pathogenic nuclei emitted by one patient's breathing and forced expulsion (cough/sneeze) under a variety of ventilation conditions. In addition to the patient, the current study includes a health care worker for the purpose of examining that worker's potential for exposure to the infectious agent. The ventilation design shown in Huang and Tsao [12] to be most successful in removing the pathogenic particles is employed here, and both the healthcare worker and the patient are heated to represent actual conditions.

2. Simulation methods

Indoor air motion is governed by the Navier–Stokes equations for conservation of momentum and by conservation of mass and have been solved using a variety of approximation and closure methods over the past 40 years. They are not provided here because they appear in numerous references [17–19]. For this study, CFD simulations were performed using the Fluent v.6.1.22 software (Fluent, Inc., Lebanon, NH), with mesh generation from the Gambit v.2.3.16 pre-processor. The Fluent software is designed to approximate the Navier–Stokes equations by a selected approximation model and then solve the set of equations for the designated boundary conditions using a finite volume method. A transient simulation was performed for studying the time evolution of nuclei transport. For these simulations, the realizable $k\sim\varepsilon$ method was employed because it has been proven to model separating flows with better accuracy than the standard $k\sim\varepsilon$ method but is not as time-consuming to run as a large eddy simulation [19].

The room geometry, modeled after the geometry used in Huang and Tsao [12], is shown in Fig. 1. The room is 5.5 m wide, 3.6 m deep, and 2.3 m high. The 2.2 m wide, 2.15 m deep, 2.3 m high bathroom within the AIIR was removed from the domain because it was assumed to be treated as a separate zone. The bed containing the patient was centered at $x = 4$ m with the back of the bed against the bottom wall of the AIIR (when looking at Fig. 1). The bed/patient was 1 m wide, 1.9 m deep, and 0.7 m high. The patient's head was an ellipsoid with a 0.001225 m^2 strip across the front to represent the nose and mouth area. This area matched that simulated by Huang and Tsao [12], although they used a cubic representation for the head. Last, the 1.62 m tall health care worker was centered at $x = 3.2$ m, $y = 0.645$ m. This height was selected based on the National Health and Nutrition Examination Survey 50th percentile female height for all ethnicities in the United States [3]. The body geometry of the health care worker approximated human geometry with an ellipsoidal head, rounded shoulders and torso, and straight legs. However, limbs were not modeled. The health care worker geometry is shown in Fig. 2.

Boundary conditions used for the airflow simulation are shown in Table 1. An air exchange rate of 8.66 air changes per hour (ACH) was employed to match the optimized ventilation conditions found by Huang and Tsao [12]. Three types of boundaries were used in the simulation. Boundary conditions for velocity inlets into the room were established for air movement under the door crack, the ventilation inlet, and the patient's mouth. As performed in Huang and Tsao [12], a grille-style inlet was used to produce unidirectional airflow into the room. Following Huang and Tsao [12], no bathroom zone was represented, so a negatively pressurized door crack to the bathroom was not included in this simulation. If a door were located behind the health care worker, classical ventilation work on slot exhaust velocities [20] and more recent CFD work