A Simplified Numerical Model for Simulating Sliding Door and Surgical Staff Movement in an Operating Theatre

Carla Balocco[1]

[1]Dipartimento di Energetica, Università di Firenze, Firenze, Italy

Abstract

This paper deals with a numerical investigation on sliding door and people moving effects on the indoor climate of a standard ISO5 class OT with an ultraclean air filter system and a total ceiling unidirectional diffuser. A simple method to analyze the effects on the OT climate by different sliding door conditions combined with crossing persons and persons with a stretcher crossing is provided. Modeling of solid objects moving is handled by an indirect method, that mainly consists in keeping into account the effects of the object's movement on the airflow without simulating the real solid objects movements in inside the numerical domains, that requires a moving mesh approach in order to be realized. The proposed procedure consists in defining some user-defined logical functions in COMSOL Multiphysics models. Those functions assume binary values that identify the portions of domain where solid objects are located (binary value 1) or not (binary value 0) at the initial time of the transient simulation. The binary value assumed by the logical functions depends on assigned geometrical coordinates for each object. In those regions fluid-dynamical properties and source terms assume specific values, determining rest conditions for fluid. Time-dependent functions then allow modification of the geometrical coordinates identifying the position of the "solid" objects during time, so that a prescribed motion law can be assigned to the moving objects. Results, obtained by transient simulations for different topical cases, show a strong influence of the surgery staff movements and sliding door opening/closing on the internal flow patterns and in static pressure variation inside the OT also. These effects are related to the operating conditions of HVAC plant system, allowing to carry-out important predictions on the effects on ventilation system working conditions. Our proposed method indicates the opportunity of simulating the influence of objects movement in a fluid -filled space using a relatively simple CFD model.