QC-06-054

Testing and Modeling of Underfloor Air Supply Plenums

Hui Jin, PhD
Member ASHRAE

Fred S. Bauman, PE
Member ASHRAE

Tom Webster, PE
Member ASHRAE

ABSTRACT

The use of an underfloor plenum to deliver conditioned air directly into the occupied zone of a building is one of the key features that distinguish underfloor air distribution systems from conventional ducted overhead systems. This paper describes the development, validation, and application of a computational fluid dynamics (CFD) model for predicting the airflow and thermal performance of underfloor air supply plenums. To provide validation data for comparison with the CFD model, a series of experiments in a full-scale underfloor plenum test facility were carried out. The results of the experiments and comparison with the model predictions are described for the major variables, including plenum airflow patterns and velocities, plenum air temperature distributions, and heat exchange between the exposed concrete slab, the underside of the raised floor panels, and the supply air as it flows through the plenum. The validated CFD model was used to perform additional simulations to investigate the impact of plenum inlet design parameters (location and airflow direction and velocity) on the plenum heat gain and temperature distribution. Implications for the design and operation of underfloor air supply plenums are discussed.

INTRODUCTION

Underfloor air distribution (UFAD) offers several potential advantages over conventional overhead systems. However, barriers exist to widespread adoption of UFAD since it is a relatively new and unfamiliar technology. One of the major technical challenges is to precisely quantify the air movement and heat transfer behavior taking place in the underfloor air supply plenum. Cool supply air flowing through the underfloor plenum in a multi-story building is exposed to heat gain from both the concrete slab (conducted from the warm return air on the adjacent floor below the slab) and the raised floor panels (conducted from the warmer room above). The magnitude of this heat gain can be quite high, resulting in undesirable loss of control of the supply air temperature from the plenum into the occupied space, sometimes referred to as *thermal decay* (Webster et al. 2002; Bauman 2003; CBE 2005). To date, evidence from completed projects indicates that excessive thermal decay can be a problem.

Due to the large number of possible plenum configurations (size, shape, number and location of plenum inlets, etc.) encountered in practice, along with the complexity of the airflow and thermal behavior, it is desirable to develop a validated mathematical model of underfloor air supply plenums. This was the major objective of the research described in this paper. To characterize the major variables that must be accommodated by the underfloor plenum model, experiments were carried out in a full-scale underfloor plenum test facility. A range of practical design and operating parameters that can affect the performance of UFAD systems were investigated. The fundamental heat transfer processes and parameters that were the focus of these experiments include plenum airflow pattern and velocity, plenum air temperature distribution, and total heat exchange between the exposed concrete structural slab, the underside of the raised floor, and the supply air as it flows through the underfloor plenum. Experimental data were collected for comparison with and validation of the numerical calculations. A computational fluid dynamics (CFD) model using a commercial software package was developed to match the full-scale test facility and finely tuned to replicate the experimental measurements. In order to demonstrate the capabilities of the validated CFD

Hui Jin, **Fred S. Bauman**, and **Tom Webster** are research specialists at the Center for the Built Environment, University of California, Berkeley, California.

plenum model, it was used to conduct a series of sensitivity studies on selected design parameters.

A review of the literature yields few references related to modeling of underfloor air supply plenums. Fujita and Tomiie (1999) developed a model to estimate the convective heat transfer coefficients between the plenum air and the concrete slab below and the underside of the raised floor panels above. However, this approach did not address the variation of the heat transfer coefficient at different locations of the plenum (due to differences in velocity) or how to extend the model to plenums with different design parameters, such as shape, size, and inlet velocities. Nagase et al. (1995) measured the cooling load of a UFAD test chamber and compared that with the room extraction rate. Data showed that the cooling load was substantially greater than the room extraction rate. The agreement was improved with an insulated access floor, which indicated that the discrepancy might be due to the heat transfer from the room into the underfloor supply plenum through the floor panel.

In this paper we (1) describe the underfloor air supply plenum test facility and the accompanying data acquisition system, (2) describe the CFD plenum model, (3) describe the validation of the CFD model by comparison with the experimental data, and (4) present a sample sensitivity analysis using the validated CFD model.

EXPERIMENTAL FACILITY

Underfloor Plenum

The underfloor air supply plenum test facility was installed in December 2000 in a university warehouse with an exposed concrete slab floor. Figure 1 shows plan and section views of the plenum structure. The plenum is 22 × 74 ft (6.7 × 22.6 m) and 1 ft (0.305 m) high. The raised floor system was constructed from commercially available floor panels and included 16 variable-air-volume (VAV) floor diffusers. The plenum occupies three bays defined by 25 ft (7.6 m) on center columns in the warehouse, with one edge bordering an exterior wall. A heating, ventilating, and air-conditioning (HVAC) system delivers supply air at a controlled temperature and volume into the underfloor plenum. The HVAC system has 2,330 cfm (1100 L^3/s) as the maximum supply airflow and 55°F–90°F (13°C–32°C) as the operable temperature control. The plenum inlet was installed at the middle of the side wall next to the HVAC system. In order to reduce the cost of instrumentation and CFD modeling, the plenum was afterwards divided into two parts by a plastic partition, depicted in Figure 1 by the dashed line. Testing and modeling were accomplished on the left section of the plenum (Figure 1). The plenum section is 22 × 48 ft (6.7 × 14.6 m), which is approximately two-thirds of the original size. As shown in Figure 1, ten diffusers fall into this part of the plenum. The gaps between the floor panels and plenum edges were taped to eliminate air leakage for purposes of comparison with the CFD model predictions. The floor panels are 2 × 2 ft × 1.3 in. (0.6 × 0.6 × 0.033 m) constructed from a welded steel outer shell filled with lightweight cementitious material. The thermal conductivity of a bare panel is 1.359 Btu·in./h·ft^2·°F (0.196 W/m·K), and that of a panel with carpet tiles is 1.002 Btu·in./h·ft^2·°F (0.144 W/m·K). The plenum is built on a structural concrete slab 10 in. (0.254 m) thick with thermal conductivity of 0.54 Btu/h·ft·°F (0.93 W/m·K).

Plenum Inlet Configurations

Preliminary calculations showed that the inlet configuration can have a significant impact on the plenum air temperature variation and heat gain. Two different inlet configurations were installed and tested to provide validation data for the

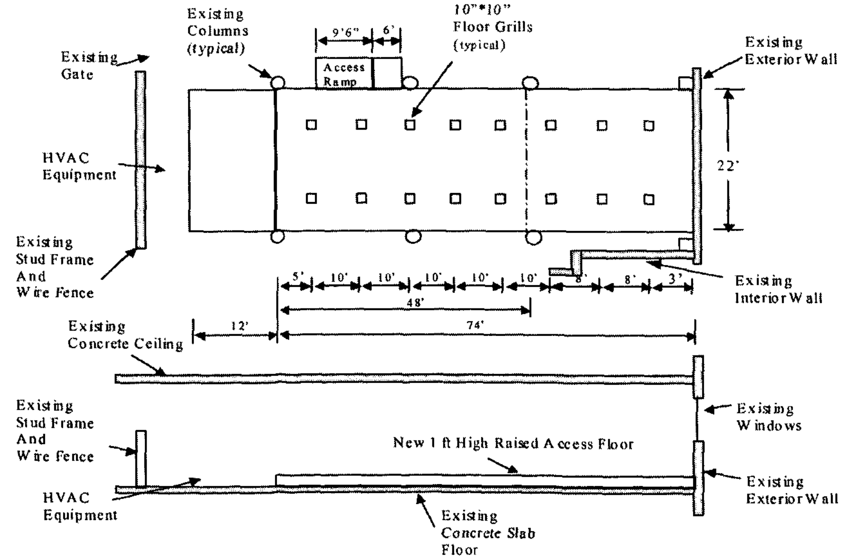

Figure 1 Plan and section views of underfloor air supply plenum test facility.

CFD model. They are one single focused jet and two jets, which is a simplified version of an inlet vane configuration that produces multiple jets. Schematic diagrams of the single focused jet and inlet vanes/two jets inlet configurations are shown in Figure 2.

Single Focused Jet. Flow straighteners and filter material are used to establish relatively uniform inlet airflow conditions, allowing more precise specification of the inlet conditions within the CFD model. It is noted that the direction of the single focused jet is not perpendicular to the short side of the plenum. The plenum inlet is located at the middle of the short side of the plenum. Thus, the geometry may be looked upon as simply symmetric. However, a symmetric geometry does not necessarily ensure a symmetric flow pattern. First, it is challenging to obtain a completely uniform inlet velocity throughout the cross-sectional area of the inlet. Second, due to the random and asymmetric nature of the turbulent flow, the airflow for the given single focused inlet jet configuration and plenum shape will not produce a perfectly symmetric pattern. In addition, information about an unrealistically symmetric flow pattern may not be normally useful in practice. In order to bypass the complexity introduced by the symmetric geometry, the direction of the inlet jet was turned to one side at a small angle from normal.

Inlet Vanes/Two Jets. Inlet vanes are one approach to spreading out and slowing down the inlet airflow. In this case, the inlet airflow can be represented experimentally as a collection of individual jets. Initial experiments and calculations indicated that the airflow pattern and air temperature distribution were very sensitive to the direction and momentum of each inlet jet. For the sake of simplicity, two jets were fabricated to investigate the influence of the inlet vanes configuration.

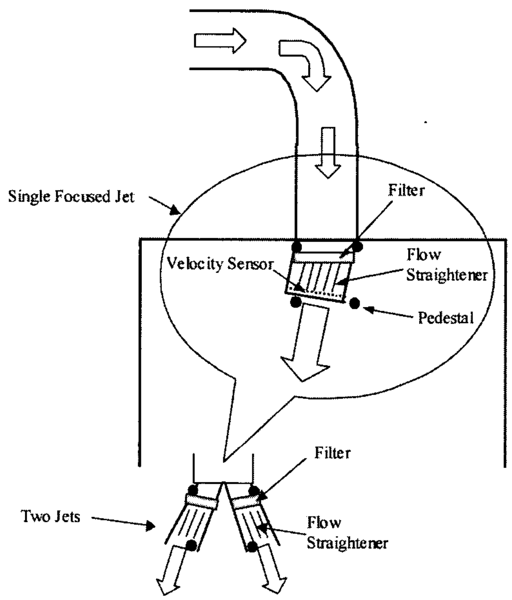

Figure 2 *Schematic of the single focused jet and inlet vanes/two jets inlet configurations.*

Measurement Setup

Type-T thermocouples and a modular data acquisition and control system were installed to monitor air temperatures at the plenum inlet, floor diffusers, and selected locations inside the plenum. Four inch (100 mm) deep holes were drilled at selected locations of the slab to obtain vertical slab temperature measurements at 1 in. (25 mm) intervals. Since there is no control of the heat transfer from under the slab (a loading dock is located on the warehouse floor below), the vertical temperature profile of the slab is very useful to implicitly derive the boundary conditions for the CFD simulation. Space air temperatures were measured just above the raised floor panels to obtain the boundary conditions above the plenum. If the heat transfer through the sidewalls of the plenum is neglected, the total heat gain into the plenum is the sum of heat transfer through the floor panels and concrete slab. In addition to convective heat transfer from the room air to the raised floor panels, recent research findings have shown that the radiative heat transfer from the warm ceiling (especially in a stratified UFAD space) to the floor could account for the majority of the heat gain through the floor panels into the plenum (Bauman et al. 2006). Therefore, the ceiling temperature needs to be considered an important part of the specification of the boundary conditions above the plenum. It was measured using an infrared temperature sensor.

The plenum inlet air velocity is measured by a low air velocity and air temperature measuring system with a cylindrical (directional) probe. The readings are then converted into the plenum inlet air supply volume by applying the cross-sectional area of the inlet opening. The air velocities at selected locations inside the plenum are measured by the same system, with a spherical (omni-directional) probe. The cylindrical and spherical probes provide the measurement range of 0.15 to 10 m/s (30 to 2,000 ft/min) and 0.05 to 5 m/s (10 to 1,000 ft/min), respectively. Both probes have the accuracy of ±3% of the readings. The voltage outputs of the anemometers are connected to the existing data acquisition system.

For the slab and floor heat flux measurements, a heat flux transducer was used. The thermal flux meter is a solid-state, flat-plate transducer designed to measure heat flux directly. The meter is placed on any surface through which the heat flow is to be measured. The heat flux meter has an accuracy of 1% of the reading. Four flux meters were attached to selected locations at the top surface of the slab and floor panel. The voltage outputs of the flux meters are also connected to the existing data acquisition system.

Figure 3 shows the locations selected for the air temperature and velocity and slab/floor heat flux sensors. Based on preliminary experimental observation and CFD simulation, the measurements at these key locations were used to characterize the predominant airflow pattern, air temperature distribution, and heat transfer from above and below the plenum.