**Title**
Using ductwork to improve supply plenum temperature distribution in underfloor air distribution (UFAD) system



**Author**
Pasut, Wilmer

**Publication Date**
2011

Peer reviewed

Sede Amministrativa: Università degli Studi di Padova

Dipartimento di Fisica Tecnica

Scuola di dottorato di ricerca in ingegneria industriale

Indirizzo Fisica Tecnica

CICLO XXIII

# USING DUCTWORK TO IMPROVE SUPPLY PLENUM TEMPERATURE DISTRIBUTION IN UNDERFLOOR AIR DISTRIBUTION (UFAD) SYSTEM

**Direttore della Scuola :** Ch.mo Prof. Paolo F. Bariani

**Coordinatore d'indirizzo:** Ch.mo Prof.ssa Luisa Rossetto

**Supervisore** :Ch.mo Prof. Roberto Zecchin

**Correlatore**: Research Specialist Fred S. Bauman

**Dottorand**o : Wilmer Pasut

*A mia moglie*

## Abstract

Cool supply air flowing through the underfloor plenum 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). These warmer supply air temperatures can make it more difficult to maintain comfort in the occupied space (without increasing airflow rates), particularly in perimeter zones where cooling loads reach their highest levels. How to predict plenum thermal performance is one of the key design issues facing practicing engineers – evidence from completed projects indicates that excessive thermal decay can be a problem.

Current research at the Center for the Built Environment (CBE) at the University of California, Berkeley indicates that unless slab insulation or other means is used to reduce plenum heat gain, one of the strategies that should be considered consists in providing the coolest supply air into the perimeter plenum zones, allowing warmer plenum temperature in interior plenums. The three approaches that have been used to accomplish this include:

- Use plenum inlets with higher inlet velocities directed at critical perimeter locations.
- Use ductwork (or DuctSox) to deliver cool air to/towards the perimeter. This is the subject of this research work.
- Instead of the typical interior plenum inlet locations, consider designing the plenum with perimeter inlets (shafts), if possible.

As described above, one of the recommended strategies for addressing thermal decay problems in UFAD systems is the use of ductwork (flexible or rigid) within the underfloor plenum to deliver cool air preferentially to perimeter zones or other critical areas of high cooling demand.

Several tests, in an underfloor plenum facility, were done to investigate the use of ductwork to improve supply plenum temperature distribution. Experimental tests have shown that using ductwork, especially vented duct with solid end cap, can cause significant pressure rising in the HVAC system.

The test results were used to calibrate and validate a CFD model. This model was compared with a previous CFD model developed and validated at the Center for the Built Environment, used by the researcher to investigate the temperature distributions in an underfloor plenum for different ways (number of inlets and inlet momentum) of supplying air into the plenum. The two models are representative of the first two strategies proposed before to reduce perimeter air temperatures in the underfloor plenum. Analyzing the simulation results comparing the models under the same operative conditions it is possible to conclude that:

- Delivering air into the plenum with a high momentum to reach the perimeter loses its capability to reduce perimeter air temperature under partial load conditions. More than this, under part load conditions (25%) there is a reversed result, i.e., the perimeter air is warmer than interior air.;
- Under the same load condition, using ductwork produces always lower perimeter temperatures;
- Delivering fresh air into the perimeter plenum with high momentum could reduce the perimeter temperatures but not the heat flux, while using a well designed ductwork reduces the heat flux and, as a consequence, the perimeter temperatures.

# Sommario

I sistemi per la distribuzione dell'aria sottopavimento (UFAD), utilizzati nella climatizzazione degli ambienti, espongono l'aria fresca che circola nel plenum allo scambio termico con il solaio (riscaldato dall'aria di ritorno nell'ambiente sottostante) e con il pavimento rialzato ( riscaldato dall'ambiente sovrastante). L'entità di questo scambio termico può essere rilevante, poiché ne consegue una di controllo sulla temperatura dell'aria di mandata (denominato Thermal Decay). L'aumento non desiderato della temperatura di mandata può creare difficoltà nel mantenimento del comfort termico nell'ambiente, in particolare nelle zone perimetrali dove i carichi termici raggiungono il loro picco.

Un lavoro condotto presso il Center for the Built Environment (CBE) ha evidenziato che, in assenza di un adeguato isolamento del solaio, dovrebbero essere prese in considerazione strategie per la riduzione della temperatura nelle zone perimetrali del plenum, anche a discapito di un aumento della stessa nelle parti interne. Le tre principali strategie utilizzate a questo scopo sono:

- immissione di aria nel plenum ad alta velocità, al fine di raggiungere le zone perimetrali dell'underfloor;
- utilizzo di condutture aerauliche(sia in metallo che in tessuto) per portare l'aria direttamente nelle zone critiche del plenum;
- considerare la possibilità di immettere l'aria nelle zone perimetrali del plenum anziché nella parte centrale.

Lo sviluppo e l'analisi del secondo punto qui esposto è l'oggetto di questo lavoro.

Utilizzando il modello di UFAD in scala 1:1, presso l'Università di Berkeley, in California, si sono potuti investigare i profili di temperatura in pavimenti rialzati dotati di tale sistema. I test hanno evidenziato fin da subito il problema dell'aumento di pressione nel sistema HVAC legato alle perdite di carico nei condotti. Il problema è ancora più marcato quando viene utilizzata una particolare tipologia di condotto che possiamo definire ventilato. Tale elemento, in tessuto nella versione analizzata, presenta lungo due lati opposti delle file di fori che permettono di distribuire l'aria in modo più efficiente.

I risultati sperimentali sono stati utilizzati per validare un modello CFD che simuli la soluzione impiantistica oggetto di analisi. Utilizzando il modello CFD appena descritto ed un secondo, testato e validato presso il CBE, che simula la prima delle strategie illustrate in precedenza, è stato possibile confrontare le due soluzioni in situazioni reali di funzionamento. Per rendere le simulazioni il quanto più possibile coerenti con un caso reale, le condizioni al contorno sono state ottenute con il programma di simulazione energetica Energy Plus. La simulazione energetica è stata eseguita per il giorno di progetto estivo, per un pavimento rialzato situato al secondo di tre piani di uno stabile uso ufficio. Come zona climatica è stata scelta la località di Sacramento, California. Si è giunti così alle seguenti conclusioni:

- mandare l'aria nel plenum ad alta velocità non garantisce la riduzione di temperatura nelle zone perimetrali in condizioni di carico parziale. La distribuzione di temperatura nel plenum, inoltre varia notevolmente al variare del carico di raffrescamento richiesto, rendendo così difficile predire le performance termiche dell'underfloor;
- dal confronto delle due tipologie di distribuzione dell'aria sotto il pavimento emerge che nel caso si utilizzino condutture aerauliche le temperature perimetrali sono, per ogni condizioni di carico, inferiori a quelle registrate nel caso di mandate ad alta velocità;
- la distribuzione delle temperature in plenum che utilizzano i condotti d'aria rimane pressoché invariata per ogni condizione di carico, facilitando così il controllo delle temperature e del comfort termico nell'ambiente condizionato;
- l'utilizzo di condutture, unito ad un'attenta analisi delle strategie di distribuzione dell'aria attraverso le stesse, consente una riduzione del flusso termico tra plenum e ambienti circostanti fino al 15% del totale.

# Contents

# 1 Underfloor Air Distribution System (UFAD)

The wording “Under Floor Air Distribution System” (UFAD) identifies a method for delivering conditioned air in commercial building. The conditioned air is delivered into the plenum that is located between a structural slab and the underside of a raised (access) floor. The air enters directly in the occupied zone through supply outlets located at or near floor level.

In terms of types of equipment used at the cooling and heating plants and primary air handling units (AHU) UFAD and Over Head (OH) systems are the same. The main difference between the two ventilation methods is how they deliver the conditioned air in the room, see Figure 1-1Figure 1-2.

All UFAD systems, even with some difference, use the underfloor plenum to deliver the conditioned air into the occupied zone through floor diffusers. The raised floor system, which forms the underfloor plenum, is typically made of 0.6 × 0.6 m floor panels constructed from a welded steel outer shell filled with lightweight cementitious material. The typical height of raised floors used in UFAD systems is 0.3-0.46 m above the concrete structural slab of the building. The underfloor plenum can be also used as a “technical space” for power/voice/data cabling services.

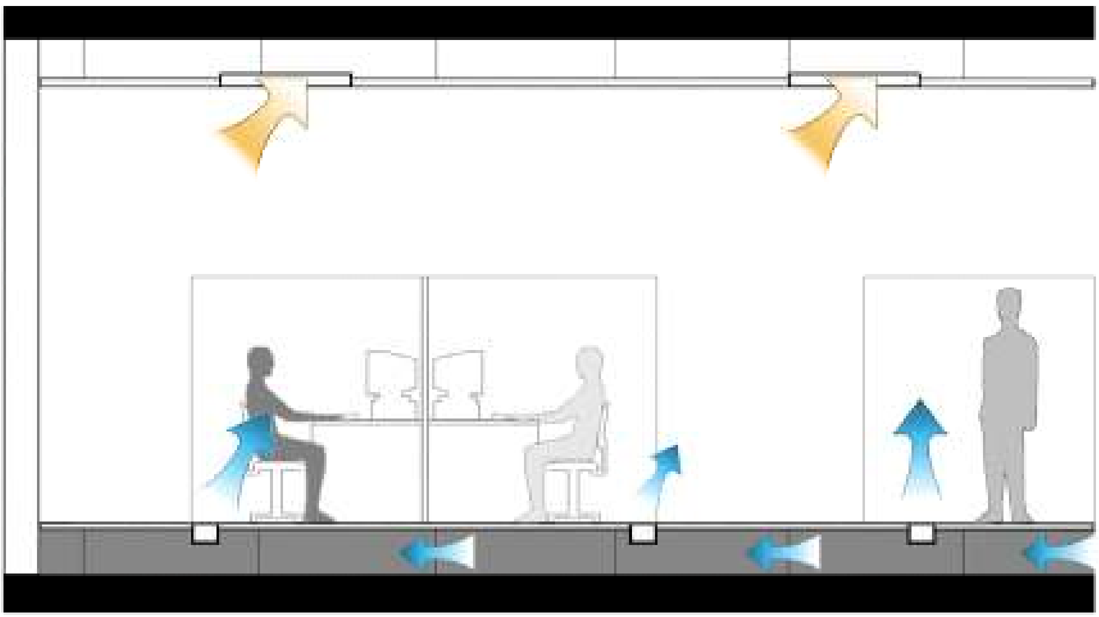


*Figure 1-1 Underfloor Air Distribution (1)*

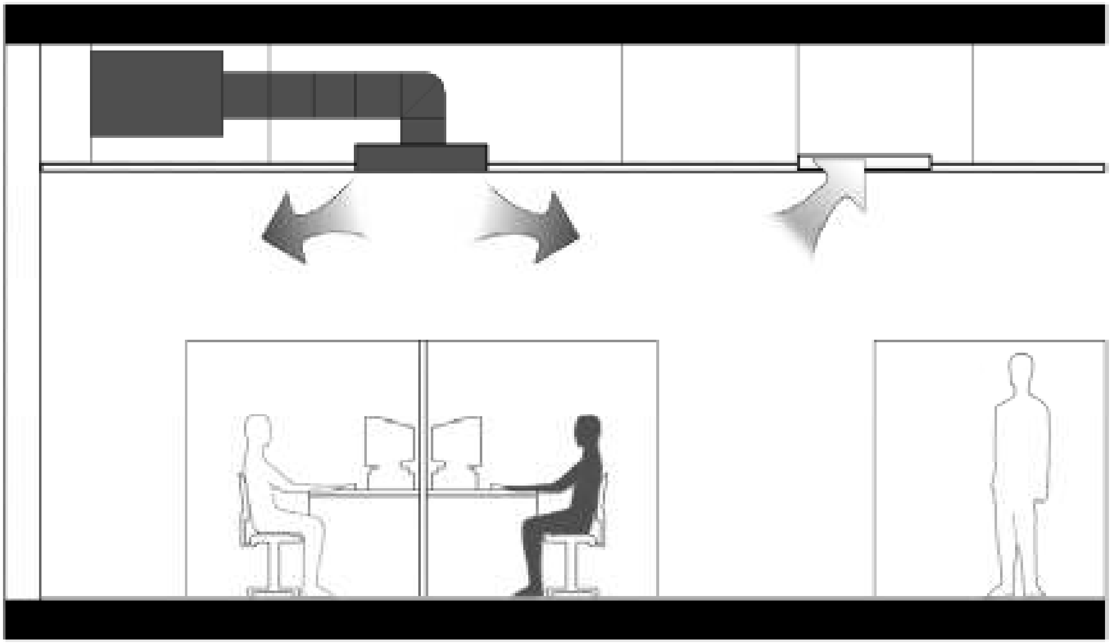


*Figure 1-2 Conventional overhead air distribution system (1)*

The basic approaches for configuring an underfloor air supply plenum are presented below:

- pressurized plenum with the conditioned air delivered from the central air hanldler through the plenum and into the space through passive grilles/diffusers, modulated diffusers, and fan powered thermal units;
- zero-pressure plenum with air delivered into the conditioned space through local fan-powered (active) supply outlets in combination with the central air handler;
- ducted air supply through the plenum to terminal devices and supply outlets(1).

In this kind of system the air is supplied directly into the occupied zone. For this reason the air floor supply outlet temperature should be maintained no lower than 16-18 °C to avoid discomfort problems due to the cool temperature close to the occupants. The number of small supply outlets close to the occupants is relatively high. The advantage offered from this kind of system is that generally the outlets close to the occupants are adjustable or thermostatically controlled , providing an opportunity for adjacent individuals to at least have some extent of control over their perceived local environment.

For the underfloor system the air return is located at ceiling level, or at the maximum allowable height above the occupied zone. This produces an overall floor-to-ceiling airflow pattern that takes advantage of the natural buoyancy produced by heat source in the office and more efficiently removes heat loads and contaminants from the space, particularly for cooling application (1). An interesting particularity of UFAD system, compared with other ventilation strategies, is that it promotes some amount of stratification in the space. This phenomenon could weight in a positive way on energy consumption and air quality in the breathing area.

## 1.1 History of Underfloor Air Distribution

The cable management and heat load removal issues caused by the proliferation of electronic equipment through the office, in the 1970s, pushed the introduction of underfloor air distribution into office building in the Western Germany (2). In these buildings, the comfort of the office workers had to be considered, giving rise to the development of occupant-controlled localized supply diffusers to provide task ambient conditioning. Some of the first UFAD systems in Europe used a combination of desktop outlets for personal comfort control and floor diffusers for ambient space control (3).

Prior to the 1990s, office installations using underfloor systems had gained primarily in South Africa, Germany, and other parts of Europe. The technology was not commonly used in North America prior to 1995, in part due to the downturn in office building construction beginning in the mid-1980s. Japan did not experience this same downturn, and as result, significant growth in UFAD technology was observed there during this period. Between 1987 and 1995, more than 250,000 $m^2$ of office space in more than 90 buildings were installed with UFAD systems in Japan (4).

However, in the late 1990s, growth for raised floor installations in the United States was dramatic, and designers and manufactures predicted that 35% of new offices would use raised floors by 2004. Half of these installations were expected to incorporate UFAD technology. This rate of increase slowed in 2003 due to the economic downturn and reduced office construction. (5)(1)

## 1.2 Applications

Generally UFAD systems are one of the best solutions for all office buildings, and even in those with an open office plans it can allow occupants to individually control their local workstation environments by adjustable diffusers. In office and other businesses with extensive use of information technologies and typically high churn rates(e.g., dot-com office, call centers, trading floors), the flexibility provided by service delivery system, including cable management, is a great benefit. Other buildings suitable for UFAD system include schools, television studios, and light manufacturing installations that do not involve spillage of liquids.(1)

In high ceiling spaces UFAD systems provide good energy-savings opportunities in cooling applications by promoting thermal stratification. Comfort and improved indoor air quality are maintained in the occupied zone near the floor, while allowing increased temperature and pollutant concentrations to occur at higher elevations in the space. Auditorium theaters, libraries, museums, and converted warehouses all make good UFAD applications. In contrast, these types of buildings can present problems with conventional overhead air distribution design(1).

## 1.3 Thermal comfort benefits

One of the greatest potential improvements of UFAD systems over conventional overhead systems is in the area of occupant thermal comfort, in that individual preferences can be accommodated. In a work environment every single person can have a different comfort preference related to the clothing (e.g. differences in terms of clothing between a men and women), activity level, and individual preferences because, as presented by Zhang et al., within a certain sensation range people's preferences are quite different(6),(7). In the Center for the Built Environment (CBE) website some interesting examples are presented: regarding clothing differences, a person with slacks and a short-sleeved shirt (0.5 clo) can perceive the same level of comfort but with 2°C less for the room temperature than a person who is wearing a business suit (0.9 clo). As an example of the variations in activity level that commonly occur, a person walking continuously around in an office (1.7 met) will experience an effective temperature of the environment that is approximately 2 to 3°K warmer than for a person sitting quietly at their desk (1.0 met), depending on clothing level (7).

The UFAD systems are involved in the design of the large majority of Task/Ambient Conditioning (TAC) systems. A TAC system is defined as any space conditioning system that allows thermal conditions in small, localized zones (e.g., regularly occupied work locations) to be individually controlled by nearby building occupants, while still automatically maintaining acceptable environmental conditions in the ambient space of the building (e.g., corridors, open-use space, and other area outside of regularly occupied work space) (1). In an office equipped with a regular underfloor system, the occupant can control his/her personal thermal comfort by adjusting operable floor air diffusers or in some cases, even the diffuser position can be changed. With a TAC system the occupant can have even higher degree of personal comfort control by adjusting the speed, the direction and in some cases the temperature, of the incoming air. Figure 1-3 presents the climatic chamber used by the CBE's research group for the thermal comfort tests. In this picture the underfloor system is combined with a TAC system.

***Figure 1-3 CBE's test chamber with TAC system***

## 1.4 Room air stratification

Another feature of underfloor air distribution system is the air stratification generated in the room. This feature can be seen as a positive characteristic of this kind of system because it helps to reduce the energy consumptions and improve the air quality in the breathing zone.

The theoretical behavior of UFAD system is based on plume theory for Displacement Ventilation (DV) systems. In a room equipped with DV system, the cool supply air is introduced near the floor at a very low velocity and it flow across the floor. It moves upward through entrainment by thermal plumes which are developed by heat sources in the room, such as occupants, computers and electrical equipment, etc. For DV systems, as well for UFAD system, it is common to identify an interface dividing the room into two zones, upper mixed and lower occupied zones, having distinct airflow conditions. This interface will be identified as stratification height (SH) (Figure 1-4 Stratification height in a room equipped with a DV system.

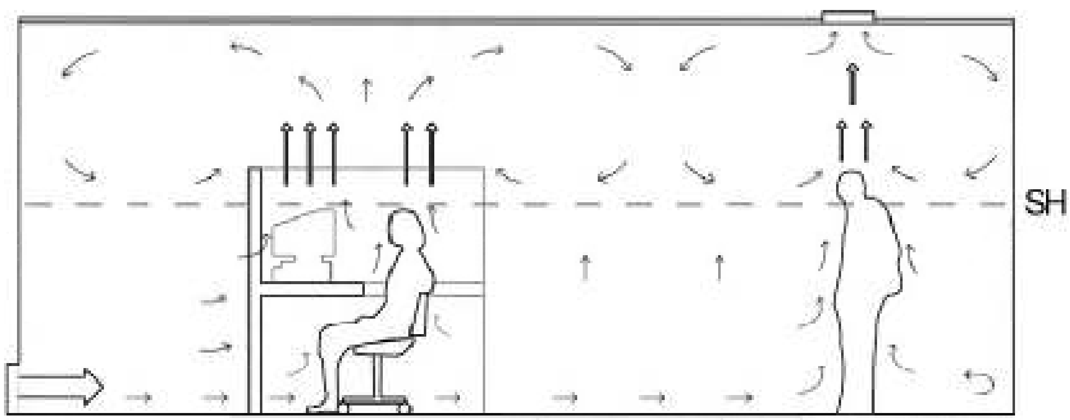



***Figure 1-4 Stratification height in a room equipped with a DV system(1)***

The lower zone, close to the displacement inlet, is the place where the stratification develops, and where there is no recirculation. In the upper zone, above the stratification height, there is a recirculating flow which produces a fairly well-mixed region. While in DV systems the height of this stratification level primarily depends on the room airflow rate relative to the magnitude of heat source, in UFAD systems, the higher air momentum inlet alters the thermal behavior of the stratified zone by increasing the amount of mixing and changing the temperature profile (8).

Figure 1-5 presents a typical vertical temperature profile for UFAD system compared with DV system and well mixed system. As it shows, based on some experimental results(9), there are up to three distinct zones to characterize the vertical thermal distribution in a room equipped with an underfloor system. In Figure 1-5, the non dimensional temperature used for the x axis is calculated with several temperatures, where T is the room air temperature as a function of height, $T_s$ is the supply temperature at the floor, and $T_E$ is the exhaust temperature at the ceiling. In the same figure another height called Throw Height is prersented. It represents the throw height of the floor diffusers. This is an important parameter to control the stratification level in the room.

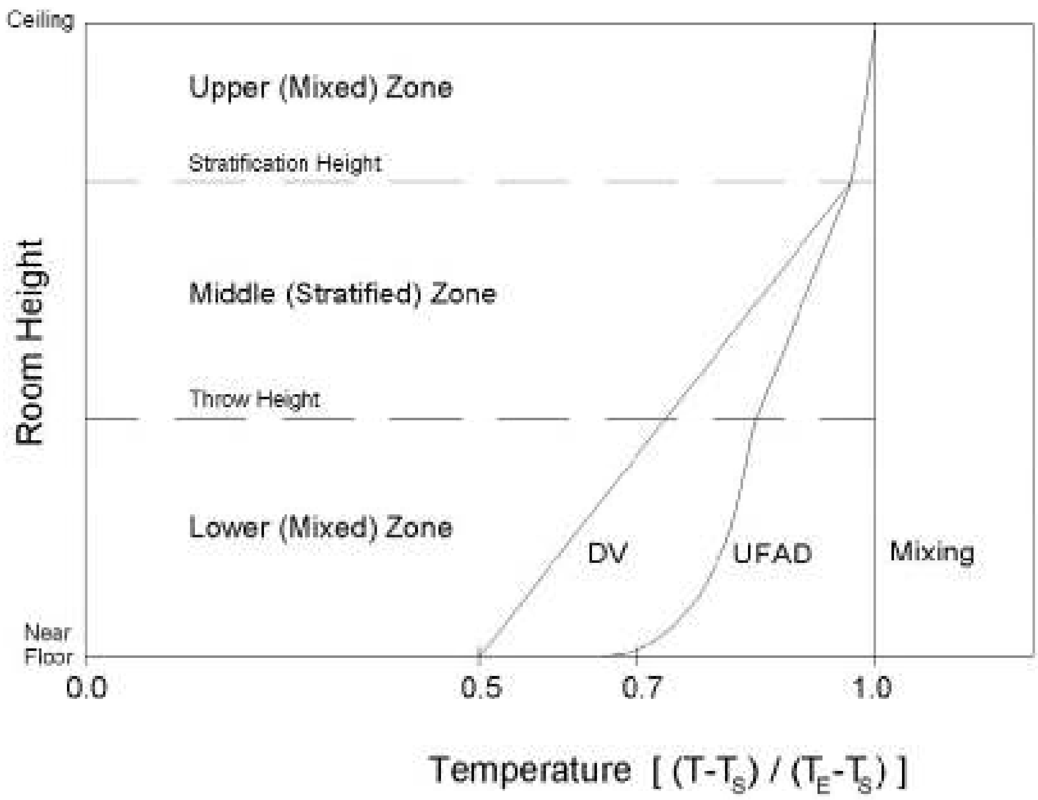



***Figure 1-5Comparison of typical vertical temperature profiles for underfloor air distribution, displacement ventilation, and mixing system(1).***

The thermal stratification must be controlled not to exceed the limit of 3 °C for the head-foot temperature difference, specified in ASHRAE STANDARD 55 (10).

## 1.5 Air Supply Plenums

The underfloor air supply plenum is one of the most significant parts in a UFAD system. The underfloor plenum allows the direct delivery of the conditioned air in to the occupied zone of the building, and it represents the biggest difference between underfloor air distribution system and ducted overhead air distribution system. A well designed underfloor plenum must supply air with the necessary quantity, temperature, and humidity, and with at least the minimum amount of ventilation air, wherever it is needed across the floor.

A typical underfloor plenum is showed in Figure 1-6. It can be defined as the open service distribution space between a structural concrete slab and the underside of a raised, or access, floor system (1). It is made by metal panels filled by cementitious materials. These panels are supported and fixed at each corner of an adjustable pedestal. Each pedestal is fixed to the concrete slab with an appropriate glue. For plenums with the greater height (above 0.45 m), horizontal stringers between pedestals and additional diagonal seismic bracing may be added.

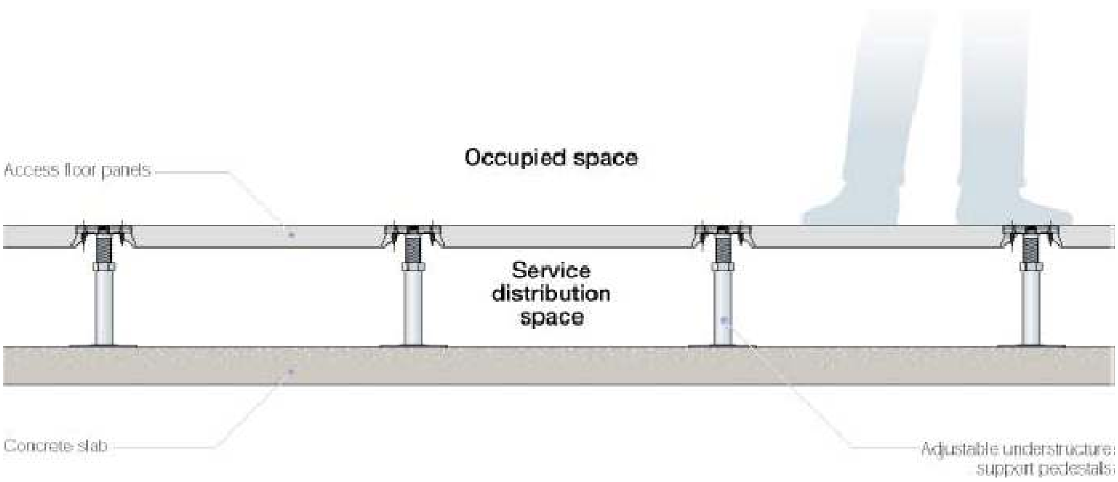



***Figure 1-6 Schematic diagram of raised floor system(1).***

Underfloor plenums were mainly used as space for PVD cabling, thanks to its ease of access to make changes to the modular cabling system. This ease of access is now available to the HVAC system.

As mentioned above in the chapter 1, and by Bauman in his UFAD Design Guide (1), there are three basic approaches to distributing air through the underfloor plenum:

- Pressurized plenum with the conditioned air delivered from the central air handler through the plenum and in to the space;
- Zero-pressure, or neutral, plenum with air delivered in to the conditioned space through local fan-powered (active) supply outlets in combination with the central air handler;
- In some cases, ducted air supply through the plenum to terminal devices and supply outlets.

The designs that are installed are often a combination of the above configurations.

### 1.5.1 Pressurized Plenum

In this configuration the pressure inside the plenum is kept a little higher than the pressure in the room controlled by the air handling unit (AHU). A typical pressure difference between the room and the plenum falls in the range of 12-25 Pa (1). The pressure inside the plenum is kept constant across every single control zone. If there is no strong air highway close to the diffusers in the underfloor plenum, the constant pressure allows the same passive diffusers in the same control zone to deliver the same amount of air. This is no longer true if the diffuser is close to an air highway. In this case the air velocity generates a lower pressure in some zones, which generates some differences in the air flow delivered by different diffusers under the same configurations but with different positions. This will be better explained later in this work.

For this configuration, without a ducted work and in presence of big controlling areas, the distance across the underfloor between the air handling unit and the farthest diffuser could be quite long. This may influence the supply temperature because across the plenum the air is exposed to the thermal exchange with the concrete slab and with the panels. This phenomenon could cause problems related to the controlling of the supply temperature.

### 1.5.2 Zero-Pressure Plenum

As it can be inferred by the name, in the zero pressure system the pressure inside the plenum is very close to pressure in the conditioned area. The overpressure due to AHU (the AHU for this systems deliver conditioned air to the plenum in the same manner as pressurized plenum) is neutralized by fan-powered supply outlets to supply the air into the occupied zone. This kind of supply outlets are called active. Even several zero-pressure plenum designs have been installed, they have not enjoyed the same amount of market penetration as pressurized system (1).

The zero pressure plenum compared with the pressurized plenum presents many advantages like:

- it can provide the accurate control of the supply airflow rate;
- the removal of floor panels will not impact airflow performance;
- there is no risk of uncontrolled air leakage to the conditioned space or outside;

but there are also many disadvantages:

- active supply outlets may have an higher cost compared to passive supply outlets;
- the large number of small fans can increase the energy consumption;
- the increase of noise level due to the small fans.

## 1.6 Underfloor air distribution (UFAD) equipment

This chapter is intended to provide a short list of the most common equipment. A more exhaustive explanation can be found on chapter 5 of *UFAD Design Guide* (1), from which the concepts of this dissertation have been taken. Every pictures in this chapter come from the presentation that Fred Bauman made during the workshop "Integrated design of HVAC system in buildings" in Venice (11).

### 1.6.1 Diffusers

Supply diffusers can be divided into two big categories, passive and active diffusers. Passive diffusers are defined as air supply outlet that let the air pass through the diffuser from the pressurized plenum to the conditioned space. The only cause of the flow through the inlet is the pressure difference between the pressurized plenum and the room and there are no fans directly connected with the passive diffuser. Active diffusers are defined as air supply outlet that forces the air to pass through the diffuser from either a zero-pressure or pressurized plenum into the conditioned space. Another distinction can be made between constant-air-volume (CAV) or variable-air-volume (VAV) diffusers. Even if many kind of diffusers, especially those positioned near an occupant's work location, are mechanically controllable (i.e. by the occupants) to some extent, only those equipped with some device for the automatic adjustment of the supply volume are classifiable as true VAV diffusers (1).

#### ***1.6.1.1 Passive swirl floor diffusers***

Figure 1-7 shows a swirl floor diffusers and how it works. These kinds of diffusers are made to provide rapid mixing of supply air with the room air up to the height of the vertical throw of diffuser. As this feature can be easily seen from Figure 1-7, the diffuser provides a swirling airflow pattern which makes easy mixing of the air from the plenum with the room air. For the diffuser installed close to occupants it allows a cert control of the amount of air. It is possible by rotating the face of diffuser (Figure 1-8), or by opening the diffuser and adjusting a volume control damper.

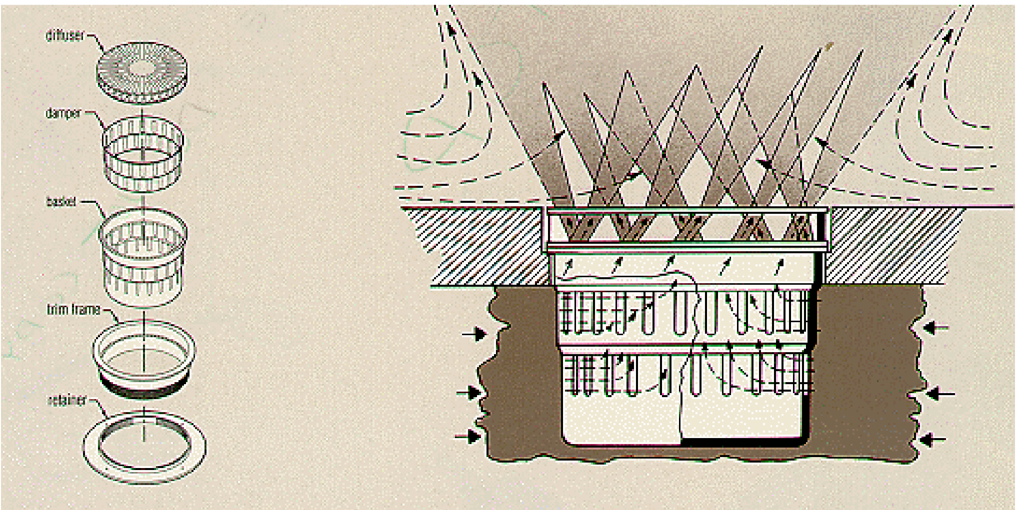



***Figure 1-7 Swirl floor diffuser***

***Figure 1-8 Air flow regulation through the diffuser***

The maximum flow rate for passive swirl floor diffusers under a typical underfloor plenum pressure of 20 Pa, is 0.04-0.047 $m^3/s$.

### 1.6.2 Passive VAV floor diffusers

Figure 1-9 shows a variable-area diffuser. The air flow pattern is different compared with the swirl diffuser. In this case it is a jet-type air flow pattern. The top of the diffuser is a slotted square floor grill, and changing the orientation of the grill allows the changing of the supply jet direction. It is equipped by an automatic or manual damper that can be used to change the air flow through the diffuser. With the damper completely opened and with a plenum pressure of 12.5 Pa, this diffuser can deliver 0.071 $m^3/s$. The particular kind of VAV diffuser presented in Figure 1-9 can perform an harmonic routine of opening and closing of the damper, which improves the air mixing between fresh air from the plenum and room air. This well mixing is obtained controlling in a coordinative way more than one diffuser at the same time.

***Figure 1-9 Passive VAV floor diffuser***

Also swirl floor diffusers can be converted into VAV floor diffusers. In this case it is necessary add a control power to automatically adjust the volume control damper.

### 1.6.3 Linear grilles for perimeter

Linear grilles are usually placed in perimeter zones adjacent to exterior windows (Figure 1-10). Because of warm façades (the main reason because they are installed in the perimeter) the grilles are connected with ducts to fan coil units to minimize the heat gain. Often these grilles have multi-blade dampers to control the air flow, but these dampers are not designed for frequent adjustment.


***Figure 1-10 Linear floor grilles***

### 1.6.4 Underfloor fan terminals

For zones where a rapid changes in cooling or heating occurs (perimeter zones close to window façades or other special zones), underfloor fan terminal are often used. Figure 1-11 shows an example of underfloor fan terminal. The one presented in the picture is equipped with two variable speed fans in order to serve different operation needs.

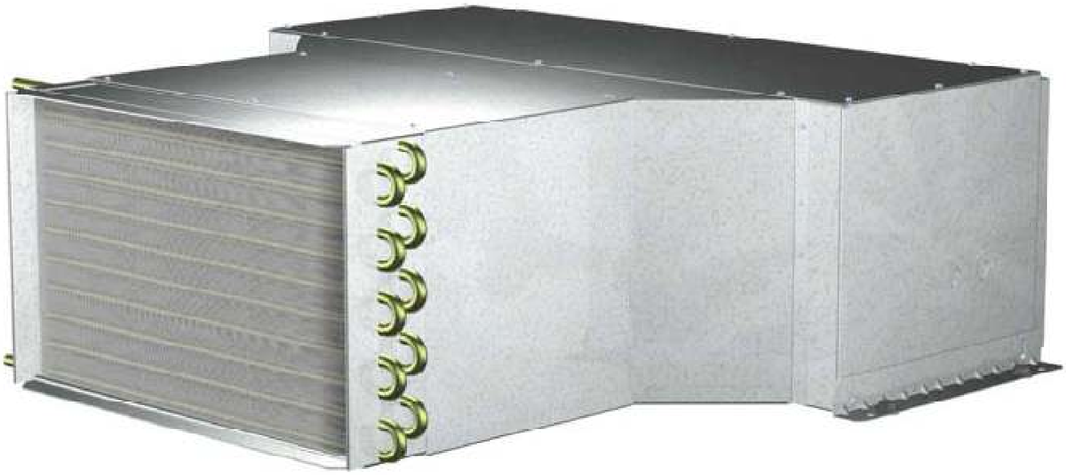
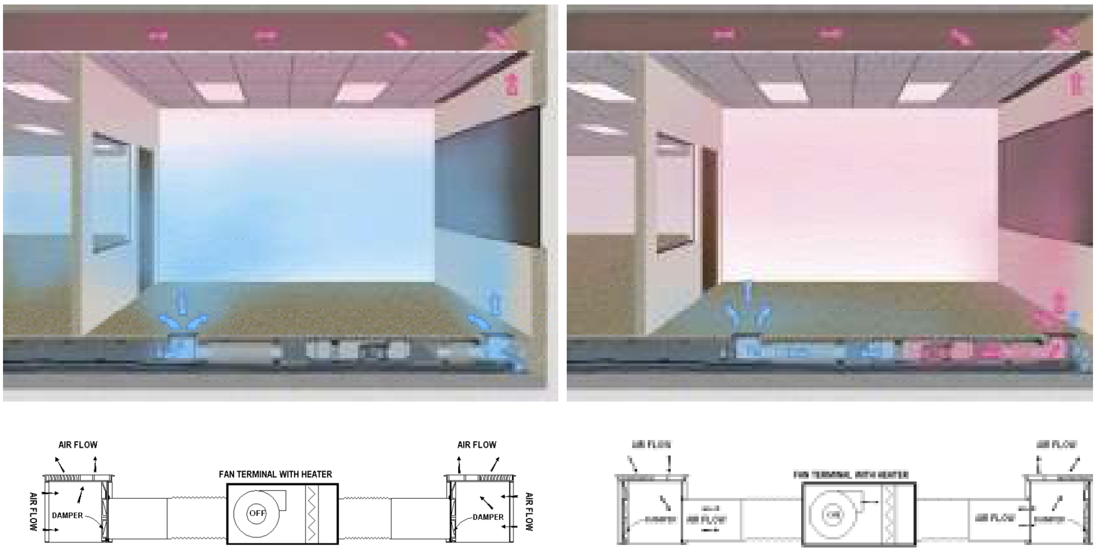


***Figure 1-11 Underfloor fan terminal equipped with two variable speed fans and hot water reheat (1)***

To serve perimeter zones two possible fan-powered configurations can be adopted. The first one is presented in Figure 1-12.




(a) (b)

***Figure 1-12 Perimeter solutions: case (a), full cooling mode; case (b), full heating mode***

The fan terminal unit, equipped with heating coil (electric or hot-water), is combined with two VAV diffusers. The systems can operate in full cooling mode, illustrated in the picture on the left, or full heating mode, illustrated on the right side of the picture. In full cooling mode in Figure 1-12(a), fresh air enters from the plenum in the two diffusers, and the dampers keep close the ways from and to the fan terminal. During the cooling mode the fan terminal is turned off. In full heating mode, shown in Figure 1-12(b), the fun terminal is turned on, and the dampers are adjusted to let air enter from the room through the diffuser on the left. The air passes through the heating coil, than it is delivered to the perimeter diffuser on the right. In this configuration room air guarantees the first stage of heating.

Another configuration to serve perimeter zones using VAV diffusers and underfloor fan units is presented in Figure 1-13. In this configuration air for the perimeter is directly drawn from the plenum, not from the room like in the configuration presented above.

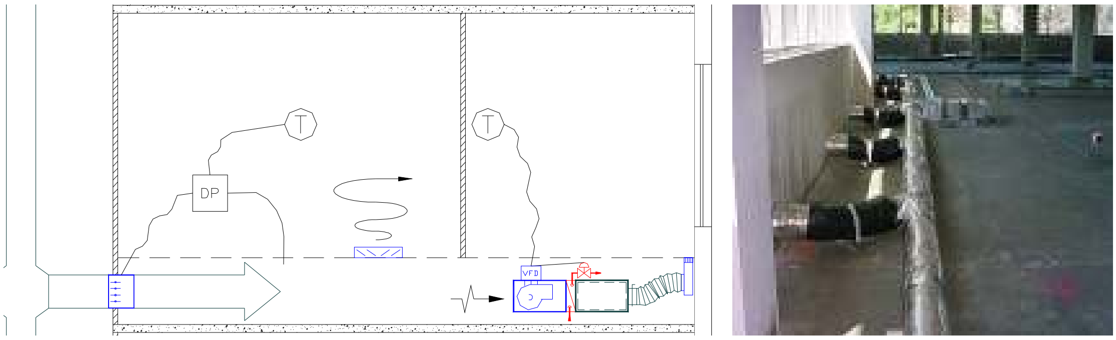



***Figure 1-13Perimeter solution which directly uses plenum air for the reheat coil***

The variable speed fan increases or decreases airflow depending on space demands. In heating the fan is set to minimum speed and the heating coil is engaged as needed. As shown in Figure 1-13, ductwork on the discharge side of the fan-coil needs to be insulated. One variation on this approach uses a small partition of the perimeter plenum instead of ductwork to connect fan-coils and perimeter grilles.

Many others configuration could be presented but giving an overview of every UFAD system configuration is not the scope of this work.

# 2 Thermal decay

One of the key-points for the proper operation of a UFAD is the room air stratification. Having a higher air stratification, anyway lower than what is prescribed for a thermal comfort (specified in ASHRAE Standard 55-2004 to be 3°C), is desirable because it leads to a reduction of total air flow and therefore to fan and chiller energy savings. For this reason a properly controlled UFAD system produces temperature stratification in the conditioned space, resulting in higher temperatures at the ceiling level than at the floor.

The definition of thermal decay is the one given by Lee K. et al. in their work (12): "thermal decay is defined as the temperature rise of the conditioned air due to convective heat gain as it travels through the underfloor supply plenum."

Thermal decay is one of the most important phenomena that have significant impact on the energy balance (12). The cool supply air flowing through the underfloor plenum is exposed to heat gain from the concrete slab (in contact with the warm return plenum from the floor below) and from the raised panels. As pointed out by Bauman F. et al. (13), the magnitude of this heat gain can be quite high, resulting in undesirable loss of control of the supply air temperature entering the occupied space. These warmer supply air temperatures could lead up to some discomfort problems in the occupied space. This phenomena could be particularly strong in perimeter zones where cooling loads reach their highest level, and the conditioned air spends more time in the plenum before reaching the perimeter diffusers(this is true in the no-ducted UFAD plenum). The thermal stratification in the conditioned space increases the thermal decay. With the increase in the stratification the temperature near the ceiling rise, and this aggravates the thermal decay phenomena. The ceiling temperature rise affects the thermal decay through the following pathways (13):

- through the radiation from the ceiling to the top of raised floor. This is the predominant part of the heating exchange, under normal conditions, through the top panels of the raised floor. In fact only a small amount of the total heat transferred through the floor surface is due by convection between the floor surface and the room air close to the floor;

- conduction through the slab from the warm return air situated in the floor below the underfloor plenum.

In the Bauman's work it is shown that up to 40% of the total room cooling load leaving the space goes into the supply plenum.

The influence of thermal decay on UFAD performance is investigated by Lee at al. (12). In this paper the whole building simulation software EnergyPlus is used to investigate the influence of thermal decay on energy consumption. EnergyPlus allowed to simulate a three-story prototype office building for three different plenum configurations. Whit this program it was possible for the researchers to model each underfloor plenum as a completely separate zone, to accurately simulate thermal decay in the underfloor plenum. The plenum area was divided into the following different zones:

- four perimeter zones (north, south, east, west), which are 28% of the total floor area (11152 $m^2$);
- one interior zone, which is 56% of the total floor area;
- one service core, which represents 16% of the total floor area.

In Figure 2-1 there is a schematic representation of the model used by Lee at al.(12). The three plenum configurations simulated are called: "series"; "parallel"; "ducted". The plenum configuration called "series" indicates that all the conditioned air leaving the air-handler first enters the interior plenum and the warmer air leaving the interior plenum then enters each perimeter plenum (case1). The "parallel" term indicates that the conditioned air from the AHU independently enters each plenum in parallel (interior plenum and perimeter plenum)(case2). "Ducted" means that the conditioned air travels through duct and thus no thermal decay exists (case3). Due to the configuration case 1 presents the highest thermal decay. For this configuration the researchers reported annual average temperature rise in the perimeter of 5.6K. This strong thermal decay leads to higher airflow to meet the zone cooling load with higher diffuser temperature. The paper shows that the thermal decay can have a big impact on the supply airflow and thus on the fan and chiller energy consumption. There is a direct correlation between the temperature rise and the supply airflow and the fan and chiller energy increase. The influence of thermal decay on UFAD performance can be summarized in the following couple of abstract from the paper (12): “When comparing case 1 with case 3 (no thermal decay), the chiller and fan energies are increased by 15% and 33%, respectively due to the thermal decay. This due to the fact that elevated diffuser temperature increases the supply airflow, which, in turn, increases the necessary chiller energy to produce the larger amount of the conditioned air. Thus, the thermal decay is related to higher cooling energy consumption. As expected, case 3 without any thermal decay shows the lowest energy consumption compared to case 1 and case 2. The annual total source HVAC energy of case 3 is reduced by 12.7% compared to case 1 with the series configuration”. This finding is very interesting because it shows that the thermal decay influencing the supply air temperature can, in turn, influence the total HVAC energy consumption. Another point related with thermal decay that must be highlighted is that the perimeter zones get the warmer supply air temperature. This is due to the solar radiation reaching the perimeter zone, especially for the west zone, and due to the warmer air coming from the interior plenum (this is only the case for the series configuration). This higher temperature in the perimeter, where the cooling load is higher, can lead to more difficulty to maintain comfort in the occupied space.

Several parameters influence the thermal decay, such as central air handler supply air temperature, zone orientation, floor level, climate, interior load and plenum configuration. The influence of these parameters can be easily understood. Only the floor level and SAT might need a brief explanation. The impact of the floor level on the thermal decay is related with the fact that the ground floor slab is in direct contact with the low temperature soil. For this reason the thermal decay in the first floor will be lower compared with the other floors where the slab exchange heat with the high temperature return air from the floor below. The impact of SAT on thermal decay is related with the airflow. In fact if the SAT increases, the thermal decay decreases mainly due to the increased supply airflow (12).

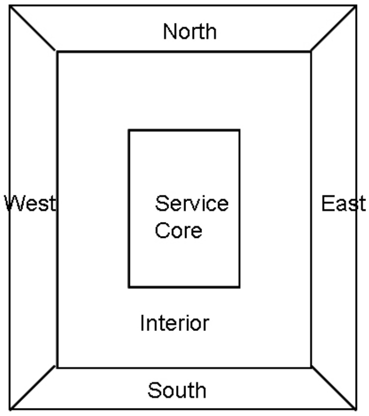



*Figure 2-1 Floor plan of the simulated building (12)*

## 2.1 Strategies to reduce or control the thermal decay

To reduce or control the thermal decay different strategies can be used. First of all a good slab insulation can reduce the heating exchange from the warm return air in the floor below the slab. Another solution is covering the ceiling, or suspended with a low-e material. This solution helps to reduce the radiation from the warm ceiling down to the top of the plenum panels. Increasing overall airflow rate will reduce thermal decay but this implies, as mentioned above, an increasing of the necessary chiller and fan energy. On large floorplates (> 25,000 ft2), adding some plenum dividers to create more control zones to control the temperature inside the plenum can also be the possible solution.

Another key point to control the thermal decay, and the room air temperature is providing the coolest supply air into perimeter plenum zones, allowing warmer supply temperatures to enter interior zones. To reach this goal few strategies were proposed these days. One of them, reported for the first time in the paper(14), is to deliver the air in the plenum with higher velocity toward perimeter. This solution helps to reduce the temperature in the perimeter air diffuser, although this benefit is counterbalanced by a warmer air temperature for the interior diffusers. This is due to the high air momentum that forces air, delivered in the plenum from the building core, to reach the perimeter plenum first and then to enter the interior plenum. This feature has a strong capability to reduce the air temperature in the perimeter zones but there is another issue to be addressed. When the system works under part load condition ( most time), the plenum air inlet velocity is not high enough “to push” the air all the way to perimeter zones. Also considering perimeter inlet location (shafts), if possible, can be a good solution to reduce the temperature rise in the perimeter.

Another common method for controlling temperature variations in the plenum is using ductwork and air highway to distribute supply air through the underfloor plenum. On the one hand pushing on the amount of the ductwork presents some interesting features to control the thermal decay and makes the thermal performance of the UFAD easy to predict, on the other hand it clutters up the plenum with ducts and partitions reducing the flexibility typical of UFAD systems.

A difference between ductwork and air highways must be highlighted. While ductwork can isolate airflow from the thermal decay, air highways can still be influenced by heat transfer from both the slab and the floor panels (1). Figure 2-2 i presents a ductwork installation for a UFAD system. As can be seen from the Figure 2-2, the presence of sheet metal ducts makes the positioning of pipes, wires and every other equipment difficult.

***Figure 2-2 Real case ductwork***

An interesting solution to avoid this problem is using ducts made of fabric instead of metal (see Figure 2-3 and Figure 2-4). The use of these squeezable ducts is very interesting for the retrofit, in which placing a metal ductwork means reconfiguring all power/voice/data cabling service, while with fabric ducts it is possible to keep the same

PVD cabling service with only few adjustment. Moreover using fabric ductwork makes the future office reconfigurations easier.

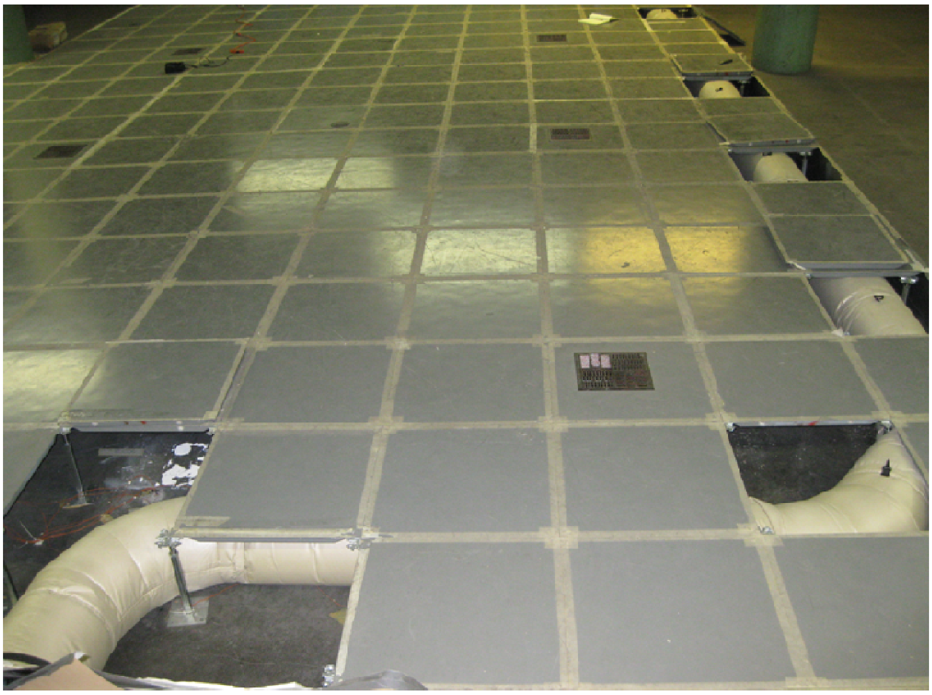


***Figure 2-3 Fabric duct in underfloor plenum***

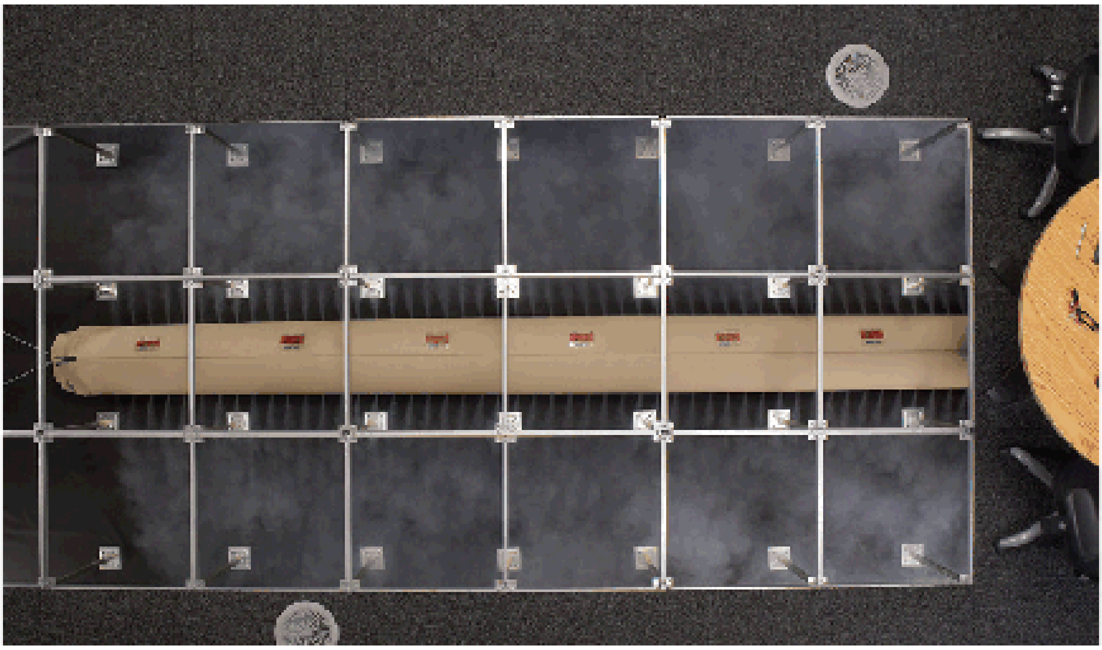


***Figure 2-4 Fabric duct in underfloor plenum (15)***

# 3 Objective

The overall objective of this work is focused on the study of ductwork (flexible or rigid) within the underfloor plenum to produce plenum air temperature distributions that are cooler in perimeter zones (where higher cooling loads exist) compared to interior zones. It is expected that this arrangement will lead to improved comfort control and energy efficiency for underfloor air distribution (UFAD) systems.

## 3.1 Background

Cool supply air flowing through the underfloor plenum 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 (referred to as thermal decay). These warmer supply air temperatures can make it more difficult to maintain comfort in the occupied space (without increasing airflow rates), particularly in perimeter zones where cooling loads reach their highest levels. How to predict plenum thermal performance is one of the key design issues facing practicing engineers. Evidence from completed projects indicates that excessive thermal decay can be a problem.

Current research at CBE (Center for the Built Environment) indicates that beside slab insulation or other means used to reduce plenum heat gain, the following strategies should be considered:

- Provide the coolest supply air into the perimeter plenum zones, allowing warmer plenum temperature in interior zones. The three approaches that have been used to accomplish this include:
  - Use plenum inlets with higher inlet velocities directed at critical perimeter locations.
  - Use ductwork to deliver cool air to/towards the perimeter. This is the subject of this research work.

- Instead of the typical interior plenum inlet locations, consider designing the plenum with perimeter inlets (shafts), if possible.

- Increase the overall airflow rate to the plenum to reduce the amount of temperature gain (thermal decay). This must be traded off against increased fan energy.
- On large floor plates (> 25,000 ft2) consider adding plenum dividers to create additional control zones.
- Consider the use of CFD simulations to more accurately predict the plenum thermal performance.
- On a ground floor with the underfloor plenum above a slab-on-grade, plenum heat gain may be reduced or changed, but careful consideration of the above strategies should still be made.

As described above, one of the recommended strategies for reducing thermal decay problems in UFAD systems is the use of ductwork (flexible or rigid) within the underfloor plenum to deliver cool air preferentially to perimeter zones or other critical areas of high cooling demand. The advantage of using ductwork is that it eliminates direct contact along the length of the duct with the slab and underside of the raised floor panels, thereby reducing temperature gain to the supply air. One disadvantage is the added flow resistance caused by the duct in comparison to airflow through an open plenum, which tends to increase the pressure drop through the system and required fan energy. In addition, once the supply air exits the duct and enters the plenum, it is still exposed to the same heat transfer from the top and bottom surfaces. The challenge and goal of this research is to determine duct layout configurations (location and number of plenum inlets) and airflow control strategies (temperature, volume, inlet velocity and direction) that provide equal or cooler plenum temperatures in perimeter zones compared to interior zones.

## 3.2 Approach

The technical approach includes the following tasks:

1) Full-scale experiments in CBE's underfloor plenum test facility.
2) Development of computational fluid dynamics (CFD) model of underfloor plenum.
3) Validation of CFD plenum model by comparison with full-scale experimental database.
4) Use of the validated CFD plenum model to conduct simulations of a broader range of plenum design and operational parameters.

# 4 Full scale experiments

A computational fluid dynamic (CFD) model, to give reliable results, must be validated with measurements collected during full scale experiments or in a case study. As described above the aim of this work is studying the use of ductwork to reduce the temperature in the perimeter underfloor plenums. For this scope the CBE's plenum facility was used to conduct a certain amount of experiments that have permitted to validate and calibrate a CFD model. A fabric ductwork was installed and studied in the plenum facility.

In this chapter, the plenum facility and the experiments made to validate the CFD models, are described.

## 4.1 Underfloor Fabric Duct

One of the approaches to reduce the air temperature in the perimeter part of the underfloor plenum is using ductwork. Either sheet metal ducts or fabric ducts can be used for this scope. For this work a plenum equipped with a fabric duct was studied.

Fabric ducts present many interesting features for underfloor plenum air distribution. Fabric ductwork features easy installation within the plenum space. The duct used during the test was made by modular and zippered straight 5ft (1.5m) long sections. This feature makes this system a good solution in cases of frequent re-design. Every module can be a solid fabric module or can present some orifices along two sides to better spread air in the underfloor plenum (Figure 4-1). The orifices can have different size depending by the airflow.

The end of the duct is closed by a modular cap. It can be adjusted to let get out a certain amount of air from the end of the duct. The modular cap can be replaced by a solid cap to completely close the duct end. Usually the diameters of fabric ducts go from 12' to 18' (0.305m to 0.45m).

***Figure 4-1 Orifices along the duct***

A certain amount of air leaks through the fabric from the duct to the plenum (see Figure 4-2). The amount of leakage through the fabric can be represented, with a good level of confidence, by a linear equation, a function of pressure difference between inside and outside the duct. In Equation 1 the correlation between pressure difference P (Pa) and leak through the fabric L ($cfm/ft^2$) is presented.

$$L = 0.0118 \cdot P$$

***Equation 1***

***Figure 4-2 Smoke visualization of air leakage through the fabric (15).***

This leakage is interesting for the scope of this work. First of all it is because it must be considered as a boundary condition for the CFD model, but also because this small leakage creates a cold layer all around the duct that avoids or strongly limits the air thermal decay from the plenum inlet to the duct outlet. This feature should be deeply investigated in future works.

Besides all the advantages presented above, fabric ductworks have also some disadvantages. The most important thing is the pressure rise in the HVAC systems related with the presence of these ducts with the end cap completely or partially closed. The pressure rise in the HVAC system results in high fan energy consumption, and in the higher air leakage in the HVAC systems.

## 4.2 Full scale plenum facility

Several full-scale experiments have been completed to investigate the influence of using a flexible fabric duct to produce cooler plenum air temperatures in the perimeter zone. To do the experiments an underfloor air supply plenum facility was used. It is installed in a university warehouse, in Berkeley, California, with an exposed concrete slab floor. The plenum area is 6.7 × 14.6 m with the height of 1 ft (30.5 cm). Originally, when the test facility was installed in December 2000, the plenum was 6.7 × 22.6 m, it occupied three bays defined by 6.7 m on center columns in the warehouse, with one edge bordering an exterior wall. The plenum was divided into two parts by plastic partitions, depicted in Figure 4-3 by a dashed line.

The raised floor system was constructed from commercially available floor panels and included 10 variable-air-volume (VAV) floor diffusers (York diffusers, Figure 4-4). The floor panels are 2 × 2 ft × 1.3 in. constructed from a welded steel outer shell filled with lightweight cementitious material. The thermal conductivity of a bare panel is 0.196 W/mK. The plenum is built on a structural concrete slab with 0.254 m thickness with thermal conductivity of 0.93 W/mK.

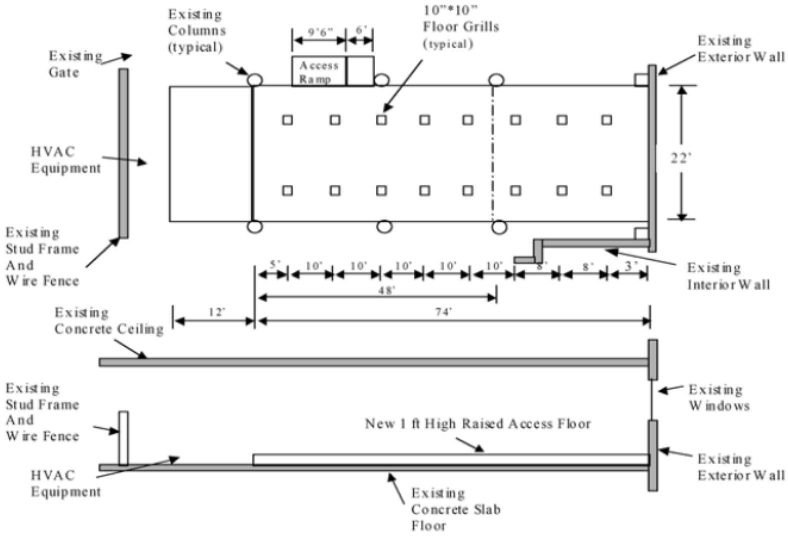



***Figure 4-3 Plan and section view of underfloor air supply plenum test facility (measures in the picture are expressed in Ft)(14)***

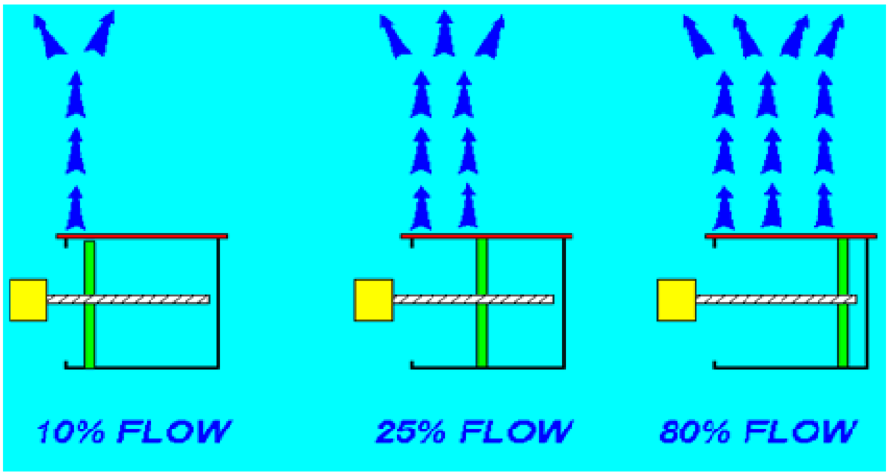



***Figure 4-4 Schematic representation of a York diffuser***

A Heating, Ventilating and Air Conditioning system (HVAC) delivers supply air at a controlled temperature and volume into the underfloor plenum (Figure 4-5). The plenum inlet was installed at the middle of the side wall next to the HVAC system.


*Figure 4-5 Plenum facility HVAC system and underfloor plenum*

## 4.3 Measurement setup

In order to collect all the data useful to validate the CFD model some measurement instruments were used.

### 4.3.1 Temperature measurement system

Type-T thermocouples and a modular data acquisition and control system (Figure 4-6) were installed to monitor air temperatures at the flexible duct inlet and outlet, floor diffusers, and selected locations inside the plenum. The acquisition system was connected to a personal computer and the data were collected with a frequency of 1 minute. Even if the most important temperatures are those measured in the floor diffusers, also the temperatures in some points inside the plenum were measured to compare the results with the temperature field predicted by the CFD program. The thermocouples inside the plenum were fixed to pedestals and positioned at equal distance from concrete slab and floor panels (Figure 4-6). The temperature measuring system was used also to measure the slab temperature. Several 0.1m deep holes were drilled in different locations on the slab below the plenum. Inside of every hole four thermocouples were embedded, one every 25 mm, and the holes were fulfilled by concrete. In this way it was possible to measure the vertical slab temperature. The average temperature measured by the deepest thermocouples inside the slab was used as boundary conditions for the CFD model.

To measure the ceiling temperature, an important parameter for the CFD model, an infrared thermometer (IR) was used.


***Figure 4-6 Data acquisition system and thermocouples inside the plenum***

### 4.3.2 Air velocity measures

An important parameter for CFD validation is air velocity. It must be measured in some selected points inside the plenum to estimate the velocity field. For this scope an omnidirectional anemometer HT-412-3 was used. The voltage output of the anemometer is connected to a National Instruments Field Point data acquisition system connected to a personal computer dedicated to this scope. A Lab View interface was used to get the average velocity value. The anemometer used for these measures is presented in Figure 4-7. It provides a measurement range of 0.05 to 5 m/s with an accuracy of ±3% of the readings.

***Figure 4-7 Omnidirectional anemometer***

To access the raised floor in the points selected for the velocity measures, some panels with a hole were used. The hole in the panels was big enough to put the anemometer into the plenum. When the hole was not used it could be closed by a cap. A cardboard was placed around the anemometer stem to close every opening during the measurement (Figure 4-8).

***Figure 4-8 Air velocity measure with the omnidirectional anemometer***

To complete the information useful to get airflow pattern inside the plenum air directions were also investigated in the same points where air velocity was measured. To do that a smoke-stick was used. When the stick is squeezed a small amount of smoke comes out from the nozzle. A kind of "cardboard window" was used to put the smoke-stick into the plenum and to avoid that air comes out from the hole. In the left picture of Figure 4-9, smoke-stick is presented, on the right it shows how smoke stick was used.

Air velocity through the orifices along the flexible duct is also an important boundary condition for the simulations. It was quite high, even higher than the range covered by the omnidirectional anemometer presented above. For this reason a directional anemometer, with the higher velocity range of 0.15 to 10 m/s, was used. Also for this anemometer the accuracy of the readings was ± 3%. The anemometer was calibrated using the wind tunnel of the Center for the Built Environment (CBE), University of California, Berkeley. Air direction through the orifices was not perfectly perpendicular to the duct axis, so it was necessary to visualize orifices air direction all along the orifices row. To do that some flexible light plastic tales were used. Figure 4-10 shows how these special tales, merged into the air flow, get oriented like the airflow.

***Figure 4-9 Air direction visualization with a smoke-stick***

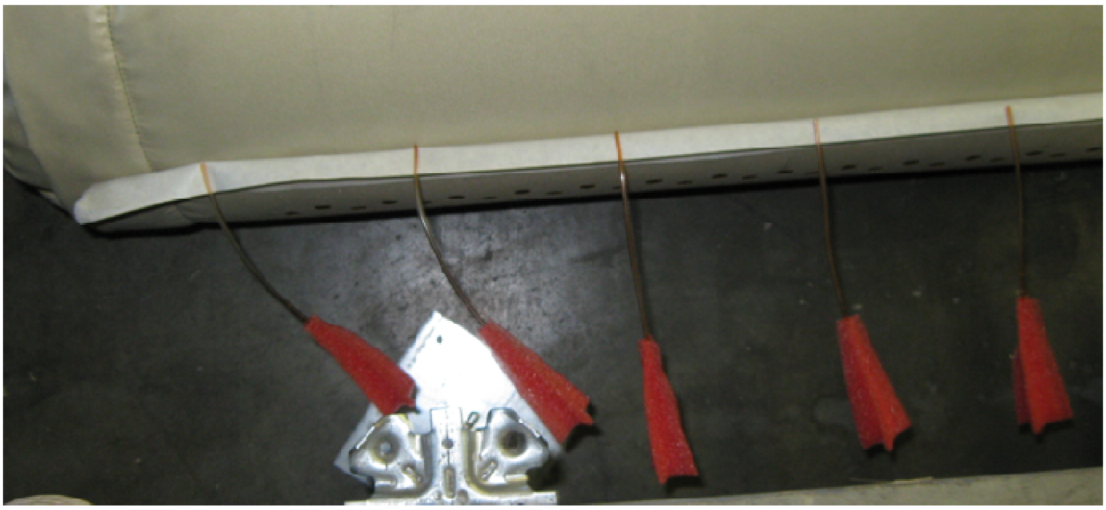


***Figure 4-10 Air direction along the orifices row***

### 4.3.3 Airflow measures

Air flow delivered into the plenum and air flow through every diffuser were measured during the tests. These important parameters to validate the CFD model were measured using a Duct Blaster. The Duct Blaster is a calibrated air flow measurement system which utilizes the difference between static pressure and dynamic pressure through a known section to get the air flow, with an accuracy of ± 3% of the readings. There are three principal components of a duct blaster (Figure 4-11), a voltage regulator (1), a digital pressure and flow gauge (2), and a fan (3), plus an element that was used to collect the airflow coming out of the diffusers (4).

(1) (2) (3) (4)

***Figure 4-11 Duct Blaster components and configuration***

Figure 4-12 shows how the duct blaster works. The blue part with a bell shape is positioned on top of a diffuser to catch the air coming out of it. The top part of the air catcher is connected to the fan through a duct. The pressure inside the air catcher is measured with the pressure gauge. The fan velocity is regulated to keep the pressure inside the bell equal to the outside pressure. In this way the airflow flowing through the fan is equal to the airflow through the diffuser without the air catcher on top. Measuring airflow through the diffusers was not only useful as a parameter to validate the CFD model, but it was also useful during the tests setup to regulate and balance the airflow in the perimeter and interior zones.

To measure the airflow delivered into the flexible duct the duct bluster was used but in a different way this time. It measured the relative pressure inside the manifold that connected the fabric duct with the HVAC system. At this point the flexible duct was removed and replaced by the fan and the duct shown before. The fan velocity was regulated to get the same pressure inside the manifold as that measured for the case with the fabric duct. In this way the airflow through the fan is the airflow entering the plenum under testing condition. In Figure 4-13 the measure of airflow entering the plenum is shown.

***Figure 4-12 Duct blaster configuration***

***Figure 4-13 Airflow plenum inlet measure***

## 4.4 Experiment configurations

Several full-scale experiments have been completed to investigate the influence of using a flexible fabric duct to produce cooler plenum air temperatures in the perimeter zone. Two different configurations were selected for comparison with the CFD plenum model predictions. Both represent a fairly simple installation of a straight length of fabric duct extending from the normal plenum inlet location on one wall down the middle of the plenum into the perimeter zone (outer 15 ft of the plenum).

### 4.4.1 Configuration #1

Configuration #1 delivered supply air into the plenum through small holes on two opposing sides of the fabric duct over the final 3 m of length (see Figure 4-14). This is a unique feature of the flexible fabric duct. The vent part of the duct could have different holes diameters and holes numbers per meter. For this test the holes diameter was ~1cm, and there were 52 holes per meter in each side. For this configuration the end cap was solid. The plenum area was ideally divided into two parts: interior zone and perimeter zone. The perimeter zone is almost one third of the total plenum area. The diffuser dumpers in the interior zone were regulated, using the duct blaster, in order to have more or less the same airflow in the interior zone and in the perimeter zone. The perimeter diffusers were completely opened.

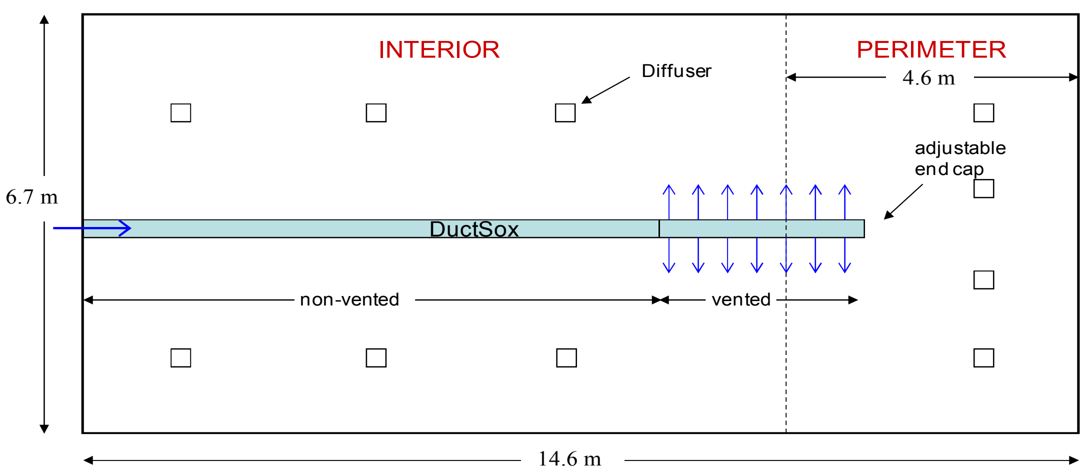



***Figure 4-14 Plan view of full-scale plenum test facility showing underfloor fabric duct configuration. Configuration #1 with vented fabric duct.***

### 4.4.2 Configuration #2

Figure 4-15 shows configuration #2, which featured unvented fabric duct with an open end. This is a useful configuration because it represents a typical duct installation (either flexible or rigid). Also for this configuration the dampers of the diffusers located in the interior part, were regulated in order to have more or less the same airflow in the interior zone and in the perimeter zone.

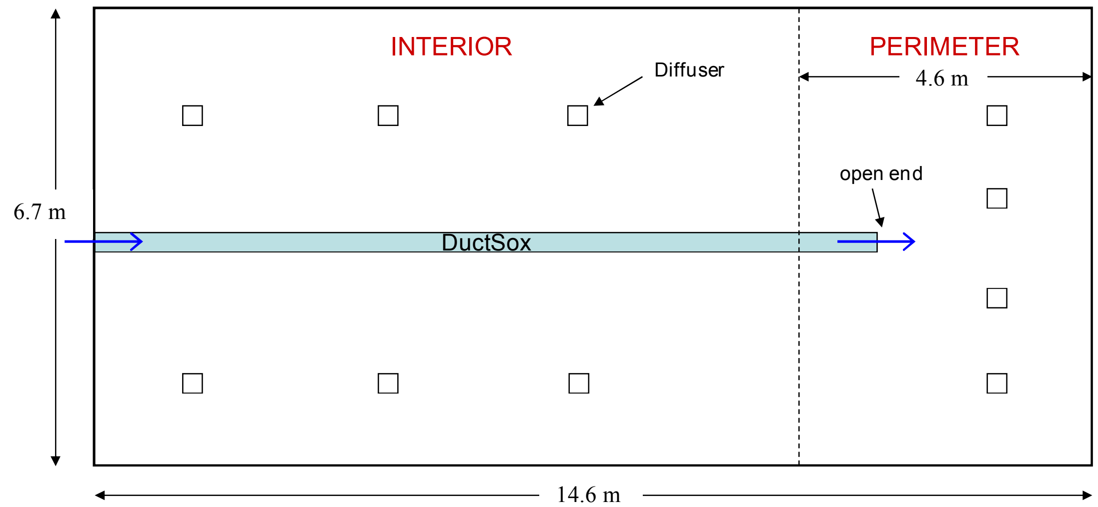



*Figure 4-15 Plan view of full-scale plenum test facility showing underfloor fabric duct configuration. Configuration #2 with unvented fabric duct and open end.*

## 4.5 Tests conditions

During the tests, the plenum supply air was measured for the two configurations. The values are reported in Table 1. Supply air reported for the configuration#1 is the maximum value reachable with the fans installed in the HVAC system due to the high pressure in the fabric duct.

*Table 1 Airflow test configurations*

| | Air inlet $m^3/s$ | Air inlet temperature [K] |
|---|---|---|
| **Configuration #1** | 0.402 | 283.76 |
| **Configuration #2** | 0.462 | 281.34 |

Outside temperature and solar radiation during the tests were not sufficiently strong to increase ceiling temperature, and given that the space below the plenum were not conditioned, the plenum supply air temperatures were quite low for both configuration #1 and #2 compared with the practice. Such low temperatures were necessary to increase the temperature differences between the warehouse ceiling and top panels surface, and between plenum air and bottom surface slab. Ceiling temperatures measured with the IR thermometer during the two tests are reported in Table 2.

***Table 2 Ceiling surface temperatures***

| | **Ceiling surface temperature [K]** |
|---|---|
| **Configuration #1** | 295.15 |
| **Configuration #2** | 296.15 |

Slab temperatures for the two configurations were measured at 4 inches depth (10.1 cm). The two temperatures are presented in Table 3.

***Table 3 Slab temperature at four inches deep***

| | **Slab temperature at 4 inch deep [K]** |
|---|---|
| **Configuration #1** | 289.7 |
| **Configuration #2** | 288.0 |

## 4.6 Test measures

The measuring system described above was used to collect all the data useful for the CFD model. The most important information are related with the diffusers. If the CFD model is able to predict the correct airflow through every diffuser and the correct temperature, it means that the model is able to catch every phenomena inside the plenum (heating exchanges, airflow resistance, air leakage, etc). For this reason the airflow through every diffuser was measured with the duct blaster. The results are presented in Figure 4-16 and Figure 4-17. As shown in Figure 4-16 and Figure 4-17, the airflows through the four perimeter diffusers are on average higher than for the interior diffusers. This is due to the dampers setting. The VAV diffusers in the perimeter were fully open, while the dampers for the interior VAV diffuser were regulated in order to have for the perimeter and for the interior more or less the same airflow.

It is also interesting to see that in configuration#2, even if the four perimeter VAV diffusers are all fully open, the two central diffusers 5 and 6 have a lower airflow compared with diffusers 4 and 7. It was verified that this phenomena is due to the strong airflow close to the two diffusers in this configuration. In fact the plenum has almost a constant pressure, but in presence of strong plenum airflows, these could form some zone with a lower pressure that could strongly influence the diffuser airflows. So far the plenum was considered as an ambient with the same pressure in every location, and so from airflow point of view the diffusers position was considered insignificant. These results show that it is not true that the airflow is only influenced by the diffuser damper and plenum pressure, but it could also be strongly influenced by the air fluxes inside the underfloor plenum.

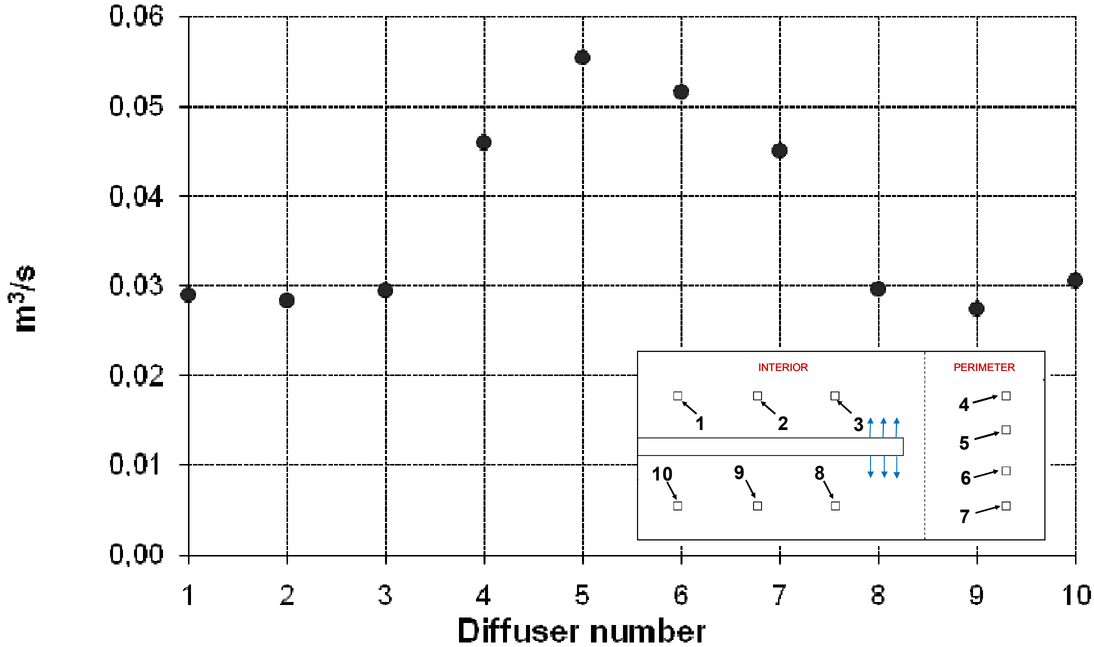



*Figure 4-16 Diffuser airflows, configuration#1*

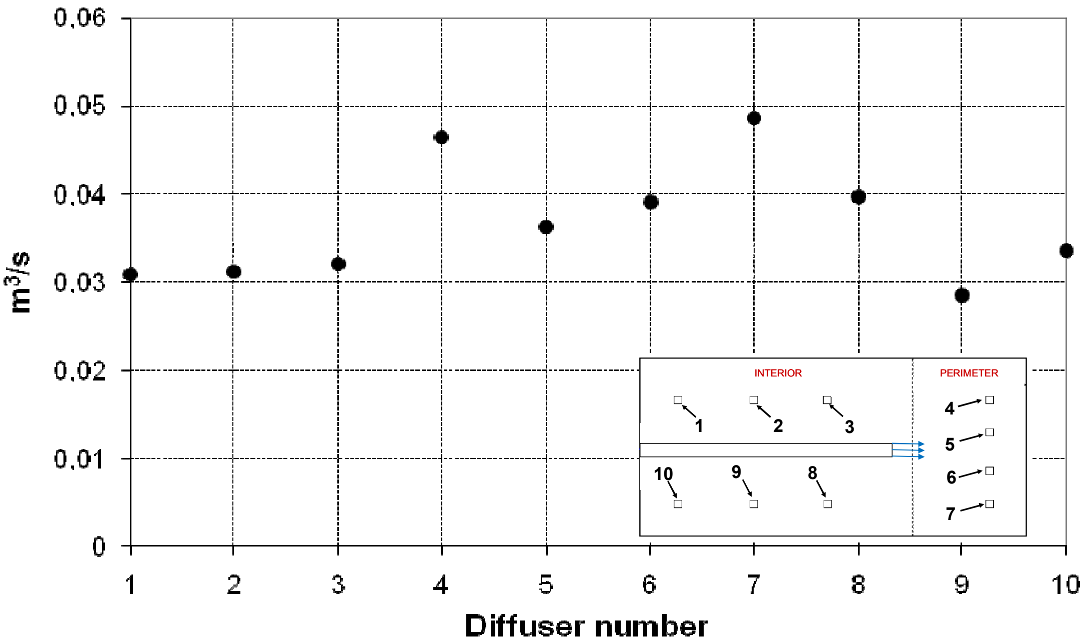



***Figure 4-17 Diffuser airflows, configuration#2***

The airflows delivered into the plenum were measured with the duct blaster and their values were compared with the sums of airflows through the diffusers. The difference between these two values is the plenum air leakage. As reported in (16) uncontrolled air leakage from the pressurized underfloor plenum (through gaps between panels, electrical floor outlets, etc.) that enters the room has a typical range for well sealed raised floors (carpet, tape, etc.) of $0.254\cdot10^{-3}$-$0.508\cdot10^{-3}$ $m^3s^{-1}/m^2$ and a typical value for not well sealed raised floors of ~ 0.0127 $m^3s^{-1}/m^2$ or greater. Uncontrolled air leakage for configuration#1 is in the range of well sealed raised floor (see Table 4), while for configuration#2 it is in the range of not well sealed raised floor. The cause of difference between configuration#1 and #2 is related with a concentrated leakage formed in the plastic partition at the end of the plenum. The concentrated leakage was caused by the strong air momentum punching the plastic partition in the configuration#2. This phenomena was modeled in the CFD plenum model because of its influence on the plenum air behavior.

The duct pressure was measured for both configuration. The result for configuration#1 is quite high compared with the practice (Table 4). Obviously the duct pressure for configuration#2 was much lower than configuration#1 because the solid cap was removed and so the fabric duct works as a regular open duct.

Another problem must be pointed out for the configuration #1, which is the duct pressure. That, other than limiting the maximum supply airflow, increased the air leakage in the HVAC system. The air leakage in the HVAC system was estimated to be one third of the total airflow elaborated by the HVAC system. This must be considered by the professional designers during the ventilation system design.

***Table 4 Test features***

| Configuration | Air inlet [$m^3/s$] | Diffuser air flow in the perimeter [$m^3/s$] | Diffuser air flow in the interior [$m^3/s$] | Plenum leakage % | %perim. %int. | Pressure in the duct [Pa] |
|---|---|---|---|---|---|---|
| **#1** | 0.402 | 0.192 | 0.164 | 11.4% | 54%<br>46% | 150-159 |
| **#2** | 0.462 | 0.170 | 0.196 | 20.8% | 46%<br>54% | 20 |

During the tests a cycling for the HVAC compressor unit was notice. This resulted in a cycling profile of the plenum supply air temperature. As shown in Figure 4-18 the compressor does an entire ON-OFF cycle every four minute. In Figure 4-18, the ordinate presents the temperature of air coming out of the duct, and the abscissa indicates the sampling number.

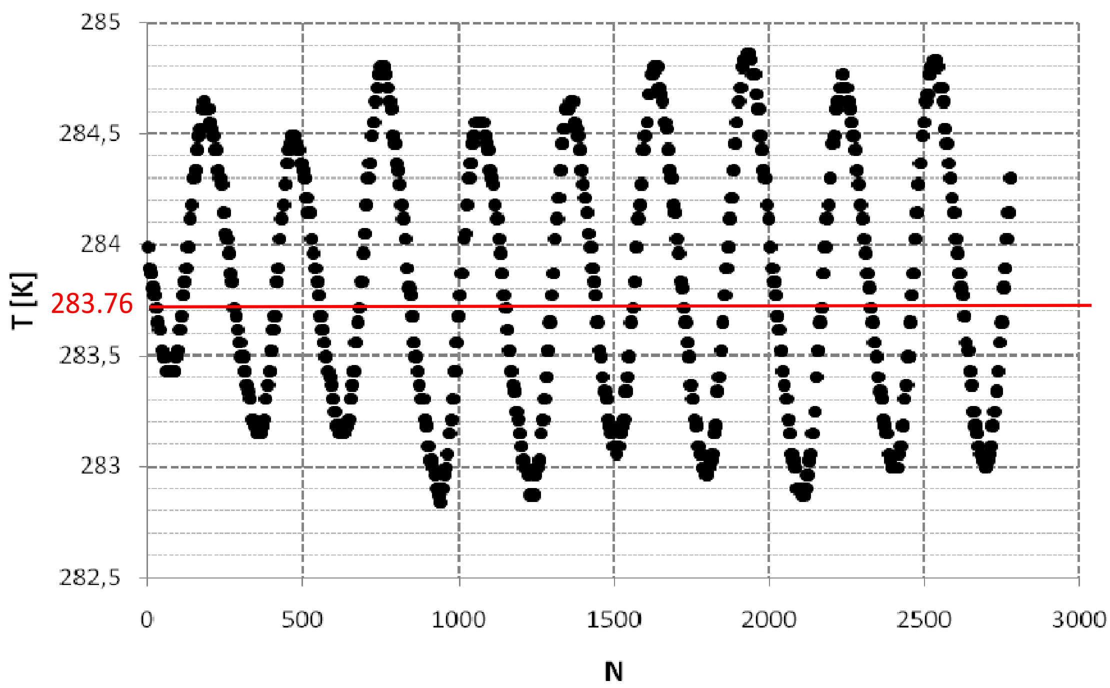



***Figure 4-18 Plenum supply air temperature cycling***

For this reason an average value is reported in Figure 4-18, which represents also a boundary condition for the CFD model. For the same reason the temperatures measured in the diffusers are an average values that were measured in the same period of time as air inlet temperature. The measured diffuser air temperatures for configuration#1 are presented in Figure 4-19. Before discussing about the values presented in Figure 4-19, it must noted that the ceiling temperature and slab temperature during this test were not so high to cause a strong thermal decay. Anyway the phenomena is still visible.

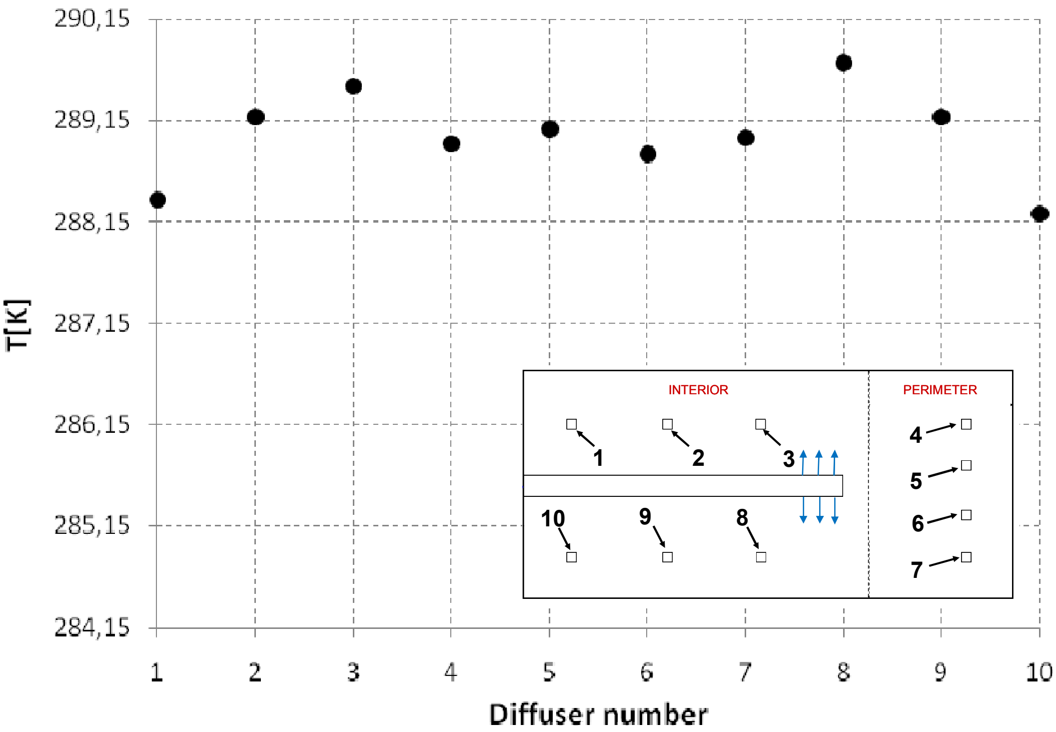



***Figure 4-19 Diffuser temperatures, configuration#1***

Also in Figure 4-20 the thermal decay phenomena is clearly visible. It is interesting to see that the airflow temperature for the diffuser number 7, located in the right corner of perimeter, is higher than the airflow temperature for the diffuser number 8. This phenomena is related with the concentrated leakage mentioned before, which strongly modifies the airflow pattern in the underfloor plenum.

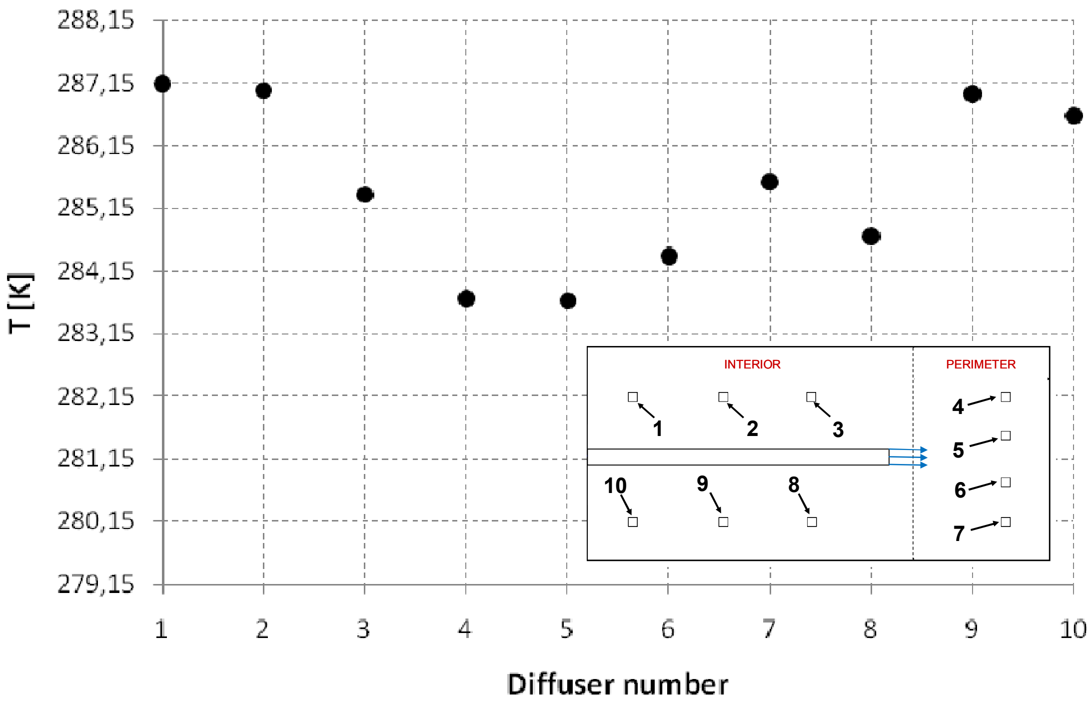



*Figure 4-20 Diffuser temperatures, configuration#2*

## 4.7 Plenum airflow pattern

Thanks to some holes located in strategic positions in the plenum panels, it was possible to measure the air velocity and to visualize the air directions inside the plenum. Considering that the plenum air flow pattern is more or less the same on the right or left part of the duct, the holes were located only in one side of the plenum. In Figure 4-21 the airflow pattern for configuration#1 is reported. The blue arrows represent the air directions in the measuring points. The air velocity, measured in m/s, is also reported. The black lines represent what should be the air flow pattern based on the smoke view of the plenum airflow. As shown in Figure 4-21, there are four big eddies inside the plenum. It is interesting to notice that, in the interior part of the plenum the eddy feeds the diffuser number one (number ten in the other side of the plenum). This is the explanation for the lower temperatures of diffuser number one and ten compared with the others diffusers, as can be seen in Figure 4-19.

The air flow pattern is presented also for configuration#2 in Figure 4-22. The meaning of symbols is the same as for Figure 4-21. In this case having the smoke visualization of the airflow pattern in both sides of the plenum should be useful, because as mentioned before the air flow pattern is not symmetric due to the presence of the concentrated leakage in the perimeter plenum edge. Anyway it is possible to assume that the fresh air coming out of the end of the duct does not involve so strongly the diffuser number seven, but when it comes down from the perimeter to the interior it involves strongly the diffuser number 8.

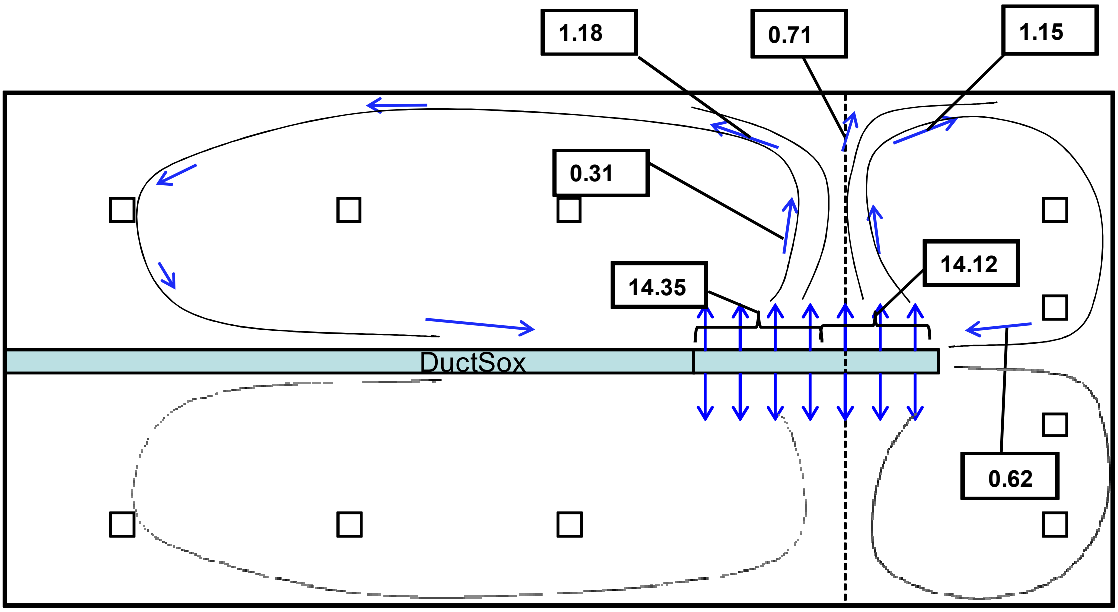



*Figure 4-21 Airflow pattern for configuration#1*

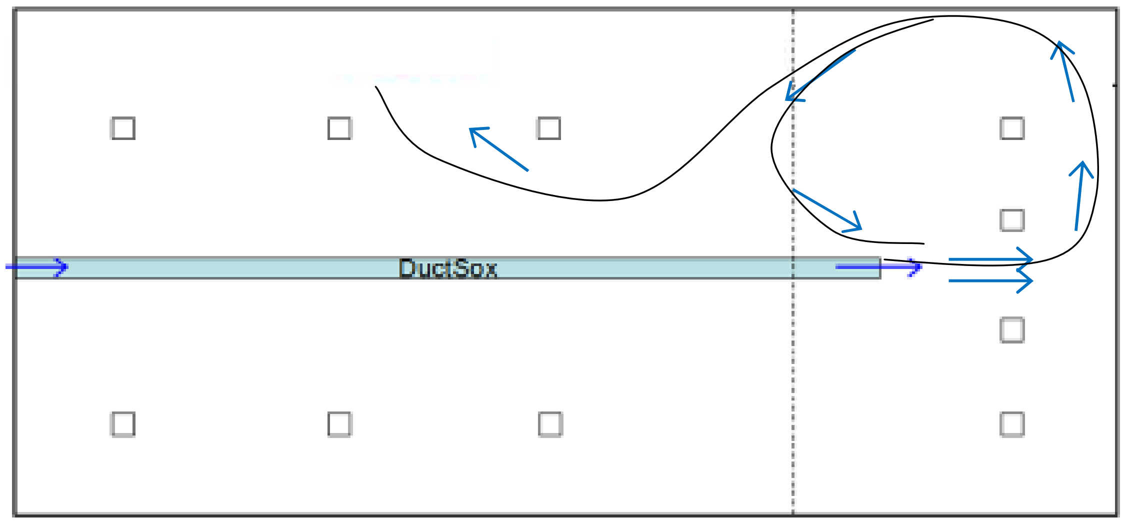



*Figure 4-22 Airflow pattern configuration#2*

# 5 CFD model

One of the best definitions of what CFD is, was given by J.Anderson et al. in their book “Computational fluid dynamics”, and it is reported here: ” The physical aspects of any fluid flow are governed by the following three fundamental principles: 1) mass is conserved; 2) *F=ma* (Newton’s second law); and 3) energy is conserved. These fundamental principles can be expressed in terms of mathematical equation, which in their most general form are usually partial differential equations. Computational fluid dynamics is, in part, the art of replacing the governing partial differential equations of fluid flow with *numbers*, and advancing these numbers in space and/or time to obtain a final numerical description of the complete flow field of interest. This is not an all-inclusive definition of CFD; there are some problems which allow the immediate solution of the flow field without advancing in time or space, and there are some applications which involve integral equations rather than partial differential equations. In any event, all such problems involve the manipulation of, and the solution for, *numbers.* The end product of CFD is indeed a collection of numbers, in contrast to a closed-form analytical solution. However, in the long run the objective of most engineering analyses, closed form or otherwise, is a quantitative description of the problem, i.e. numbers”.

The goal of this work is to verify if using fabric ductwork can reduce thermal decay in underfloor plenum, and if it can help to reduce the diffusers air temperatures for the perimeter plenum. CFD was an essential instrument to reach the objective of this work. The CFD model gave us many information about the influence of fabric ductwork on thermal decay, and it was very useful for comparing ductwork with other technical solutions used to reduce thermal decay. The CFD model was developed based on the CFD plenum model realized by Hui Jin et al. (14), but it was necessary to change many features because Hui’s model was made for a different plenum air inlet configuration. To characterize the major variables that must be accommodated by the underfloor plenum model, the experiments carried out in the full-scale underfloor plenum test facility, described above, were used. The experimental data were collected for comparison with the numerical calculations. The CFD model of an underfloor plenum equipped with a fabric duct was developed using a commercial software package. To be validated it had to match the airflow pattern, and plenum air temperature distribution.

This work presents 1) the CFD model, with special attention to describe every important parameter for the simulation; 2) the validation of the CFD model comparing the CFD results with the experimental results; 3) a new CFD model, based on the two validated models, will be compared with a CFD model developed by the Center for Built Environment (CFD), used to study a different strategy to reduce thermal decay.

## 5.1 Description of the CFD model

The use of CFD modeling to solve flow and heat transfer problems is increasing within the HVAC industry. Several papers were published using CFD simulations to address the problem of ventilation in enclosed environment (17)(18) and to compare ventilation systems (19). There are two basic approaches that can be used to study a new ventilation strategy, or ventilation system: 1) experimental measurements, 2) and computational fluid dynamics. For sure experiments can give more realistic information concerning the ventilation system capabilities, air distribution and air temperatures. This makes measurement the base of research and it is still used to evaluate performance of ventilation systems. On the other hand using measurement to evaluate ventilation systems under many different configurations and operative conditions makes this approach extremely expensive, from both economic and time point of view. For studying an air ventilation system a quite big test facility must be used and this is often a problem for researchers. If a case study can help to get around the problem, its fixed operative condition is a limitation, because that makes the measurements representative only of that case study.

Another way to proceed in the studying of a ventilation system is using CFD simulations as an extension of the experiment if these simulations are well calibrated by the experimental data. CFD simulations give many detailed information with a reasonable time and cost effort (19). This second approach was the strategy used for studying the application of fabric ducts in underfloor system. So the first step was to develop a CFD model based on the dimensions of the CBE's plenum test facility. The model had to replicate the dimensions of plenum facility, with a certain necessary amount of simplifications.

### 5.1.1 Governing equations

The partial differential equations solved by the CFD code are the unsteady Navier-Stokes equations in their conservation forms. The instantaneous equations of mass, momentum, and energy conservation can be written as follows:

The continuity equation (Equation 2):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0$$

***Equation 2***

where ρ is the density [$kg/m^3$], and *U* is the velocity vector [m/s].

The momentum equation (Equation 3):

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \times U) = \nabla \cdot \{-p\delta + \mu[\nabla U + (\nabla U)^T]\} + S_M$$

***Equation 3***

where μ is the dynamic viscosity [mPa·s], and $S_M$ is the momentum source [$kg/m^2 \cdot s^2$]

The energy equation when the contribution of the kinetic energy to the total energy can be neglected is (Equation 4):

$$\frac{\partial \rho h}{\partial t} + \nabla \cdot (\rho U h) = \nabla \cdot (\lambda \nabla T) + S_E$$

***Equation 4***

where *h* is the specific static enthalpy [J/kg], λ is the thermal conductivity [W/m·K], and $S_E$ is the energy source [$kg/m \cdot s^3$].

### 5.1.2 Turbulence model

One of the most prominent turbulent models, the *K-ε* model, has been implemented in these simulations. It is a commonly used model and is suitable for a wide range of applications. For these reasons it is considered the industry standard model. Zhang et al. in their paper “Evaluation of Various Turbulence Models in Predicting Airflow and

Turbulence in Enclosed Environments by CFD: Part2-Comparison with Experimental Data from Literature" have stated that the *K-ε* model shows the best overall performance compared to the other models in terms of accuracy, computational efficiency, and robustness (18).

*K* represents the turbulence kinetic energy and is defined as variance of fluctuations in velocity. *ε* is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate).

The transport equations for the turbulence kinetic energy and turbulent dissipation rate are presented below:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho U k) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + P_k + P_{kb} - \rho\varepsilon$$

***Equation 5***

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho U \varepsilon) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right] + \frac{\varepsilon}{k}\left[C_{\varepsilon 1}(P_k + P_{\varepsilon b}) - C_{\varepsilon 2}\rho\varepsilon\right]$$

***Equation 6***

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, $\sigma_k$ and $\sigma_\varepsilon$ are constants, while $P_{kb}$ *and* $P_{\varepsilon b}$ represent the influence of the buoyancy force. For more details please refer to (20).

## 5.2 CFD model features and boundary conditions

The main dimensions of plenum model are same as main dimensions of plenum test facility: 6.7 × 14.6 m, and 1 ft (30.5 cm) height. The raised floor height includes the plenum height (27.2 cm), and panel thickness (3.3 cm).

Jin's model of UFAD plenum (14) is modeling only the underfloor plenum, while the space above the plenum was not modeled. To take into account the thermal radiation from the ceiling to the top-face of panels an equivalent surface temperature and thermal resistance for the top layer of the model was created. This simplification introduces some approximations, which are not always acceptable. For this reason the room above the plenum is included in the CFD model for this work, the. Studying the room temperature distribution is not a scope of this work. The only benefit of modeling the room space was to simulate the ceiling radiation.

In Figure 5-1 the CFD model of configuration#1 is presented.

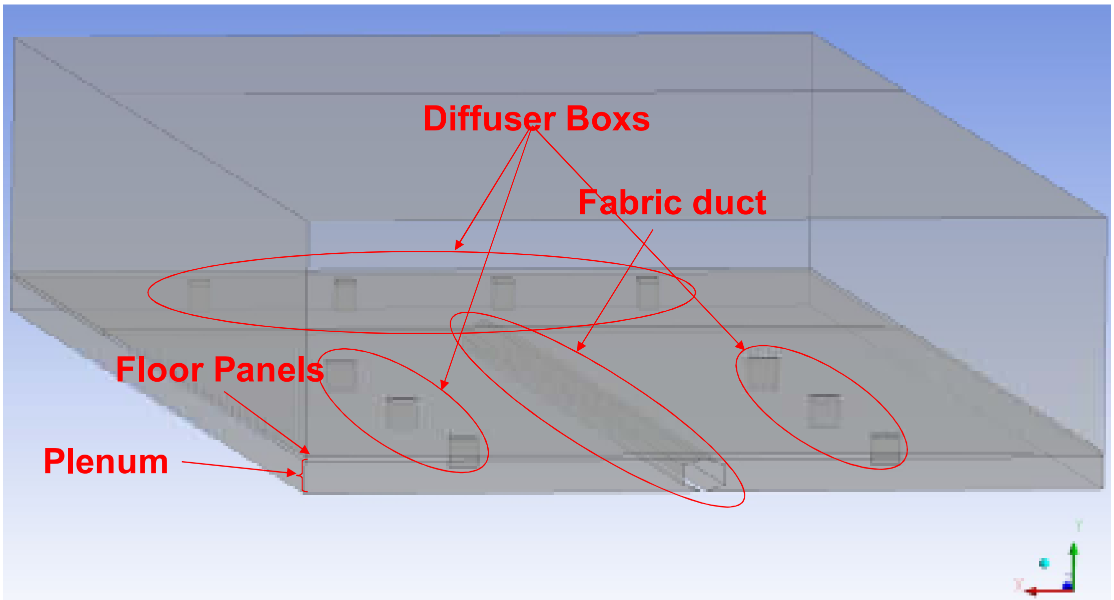



***Figure 5-1 CFD model. Configuration#1***

### 5.2.1 CFD diffusers model

Particular attention was paid during the diffusers modeling. The first simulations, made to better understand the problem, have evidenced the importance of modeling the boxes located below the diffusers. In fact if the air flow pattern is characterized by high air velocities, the diffuser boxes work as “flow-catcher”. The experiments and the simulations have shown that the plenum pressure influences not only the airflow through the diffusers, but also the airflow pathway into the plenum.

The grills placed on top of the diffusers were not modeled. An equivalent area was considered instead of the diffuser grill not to add useless details to the model. The equivalent area is not the same for every diffuser, to take into account of the damper position (see Figure 5-2).

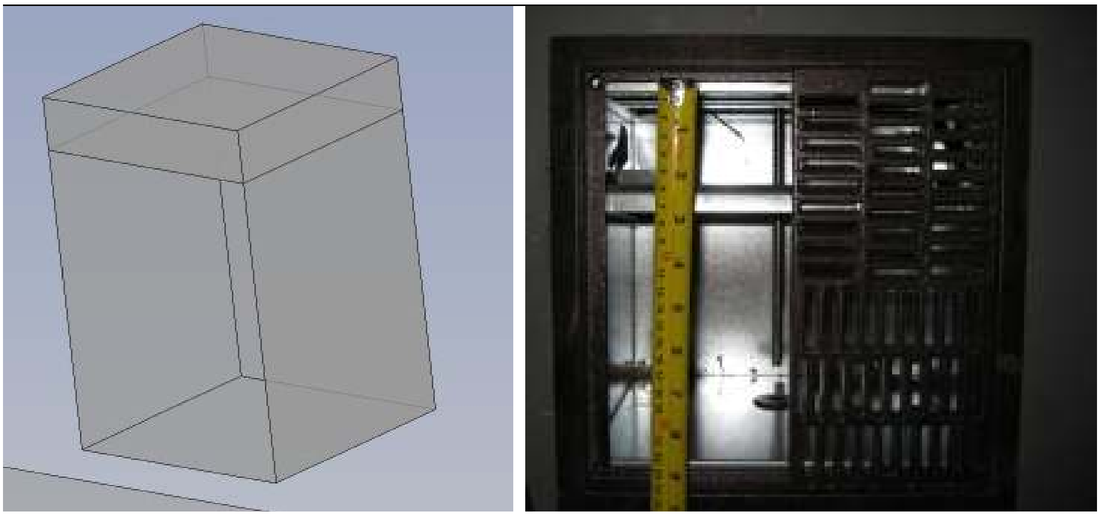


***Figure 5-2 Diffuser CFD model***

In Figure 5-2 the diffuser CFD model, and the grill located on top of the diffusers, are presented.

### 5.2.2 CFD model of fabric duct

Modeling the duct with a circular shape introduces some useless complications, like meshing complications, so the duct was modeled as an octagonal shape. The duct model has the same surface area as the real duct, in order to model the air leakage through the fabric with the right velocity.

The boundary condition option used, in the CFD program, for the duct surface was “air inlet”. The main features of this boundary condition are:

- it is possible to fix a mass flow inlet value or air velocity inlet value;
- it is possible to fix a temperature for the air inlet;
- a turbulence intensity for the air inlet can be defined;
- through a surface defined as “air inlet”, air can travel only from the outside into the model, not vice versa.

This last feature is very important. In the real case air goes only from the inside of the duct to the outside (in the plenum).

Using the Equation 1 was possible to estimate the air leakage through the fabric per square-foot, for both for configuration#1 and #2. These values were used as boundary conditions. The air velocity values through the fabric were 0.0106 [m/s] and 0.0012 [m/s], respectively for configuration#1 and #2. The air velocity is much lower for configuration#2 due to the low pressure inside the duct for this configuration.

The orifices along the two sides of the fabric duct, for the configuration#1, were not modeled as circular holes to reduce the mesh complexity. The orifices were modeled as rectangular shape inlets (see Figure 5-3). Every rectangular inlet represents more than one orifices ( a number that goes from five to twenty), and has a variable length depending on the number of orifices that represents. During the tests the air velocity was measured for every orifice, and measures have shown that its value changes along the duct. For this reason contiguous orifices with the same air velocity were bunched together in the same inlet. Every rectangular inlet has the same area of the orifices that it represents.

During the tests the air temperature was measured for the first and the last orifices along the duct. The measured temperature was the same for both positions, so the air temperature is the same for every orifice, and the measured temperature was used as a boundary condition for the air inlet in the CFD model. The temperature for the air entering the duct from the HVAC system was measured, and the difference between the air entering and coming out from the orifices is negligible. For this reason the same temperature was used as boundary conditions for the air inlets along the duct and for the air leakage through the fabric. The air inlet values for the two configurations are presented in Table 5.

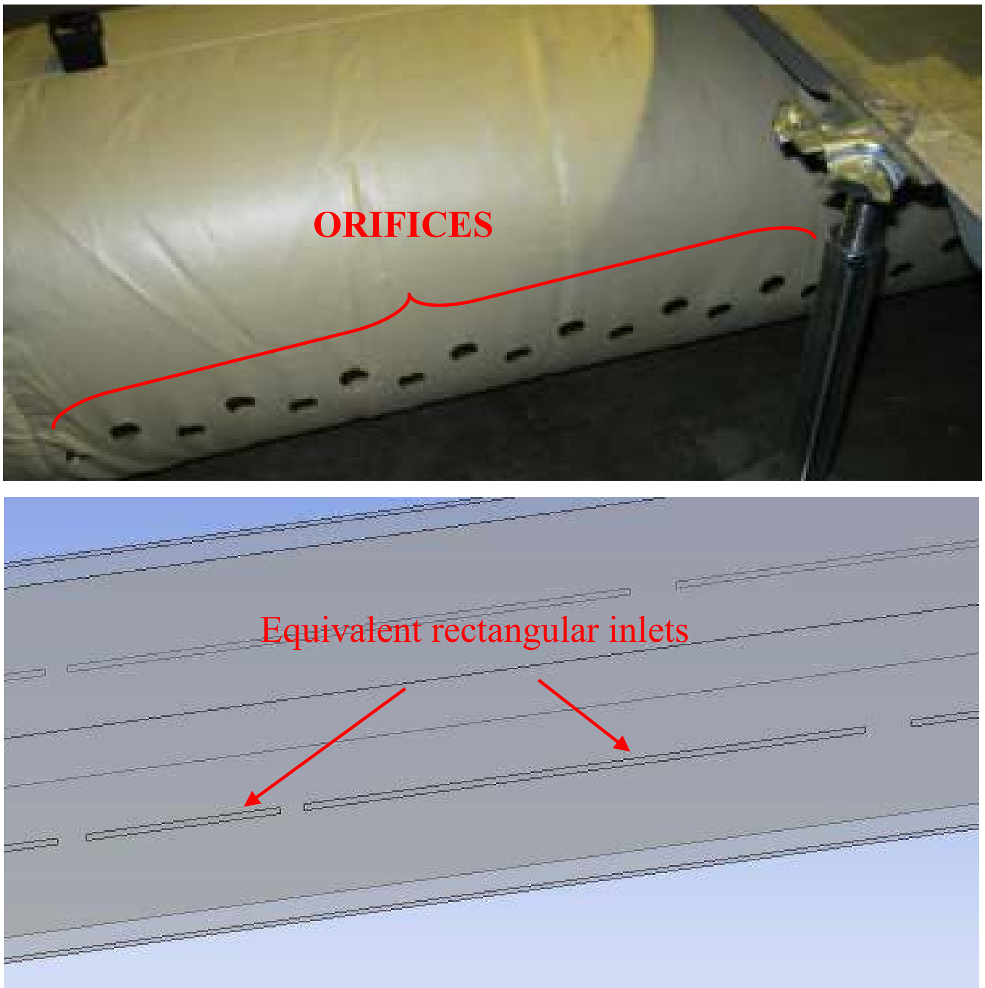



***Figure 5-3 Equivalent rectangular inlets***

***Table 5***

| Configuration | Inlet temperature [K] |
|---|---|
| #1 | 281.34 |
| #2 | 283.92 |

The air inlet velocities through the orifices were quite higharound 13 [m/s], and also the turbulent intensity was very high, estimated to be 50%.

The duct CFD model for configuration#2 has the same shape as configuration#1, but in this case more attention was paid to model the end part of the duct with exactly the same area as the real case. This was necessary to reproduce, in the CFD model, the same air inlet velocity as the real case. The air velocity inlet through the end of the duct was 5.73 [m/s], and the turbulent intensity was considered medium, which means 5%.

### 5.2.3 CFD model of structural slab

The structural slab was modeled as a single layer with an equivalent heat transfer coefficient. The boundary condition option used was "wall". When this boundary condition is used, an outside temperature and the heat transfer coefficient (HTC) must be fixed. The HTC has the scope to simulate the wall conductance. As outside temperature was used an average value for the temperature measured with the thermocouples located at ten centimeters depth in the slab. The thermal conductance was estimated for ten centimeter of concrete slab with a conductivity of 1.5 [W/m·K]. Its values is 9.2028 [w/m$^2$K]. The outside temperature was 289.70 [K] and 287.98 [k] respectively for configuration#1 and #2. Figure 5-4, taken from the "CFX help release 12", represents how the boundary condition "wall" works.

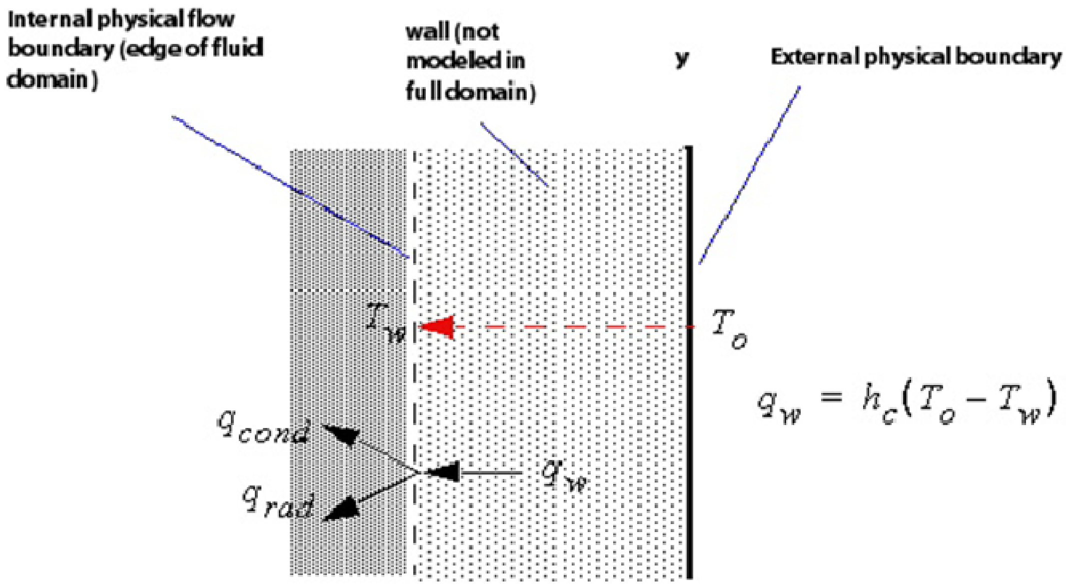



*Figure 5-4 Heat transfer for the boundary condition "wall" (21)*

### 5.2.4 CFD model of floor panels

The floor panels in the CFD model, like in the real case, represent the separation between the underfloor plenum and the room. It was not possible to model those as a single no-thick layer like the slab, but they were completely modeled with the same thickness (3.3 cm) and the same thermal conductivity (0.188 [W/m$^2$·K], given by the constructor) as the real case. In Figure 5-1 it is possible to see the panels thickness in the CFD model.

### 5.2.5 Air plenum leakage configuration#1

During the tests the air plenum leakages were measured. In the real case air leaks from the plenum to the room through the fissures between the panels. This happened also during the tests, in the plenum test facility, but to this phenomenon must add the leakage through the plenum edges. The plenum is located in the middle of a warehouse. For this reason some sheet metal walls were necessary to close the plenum edges. Even if well done these edges are not completely sealed, so a part of the total air plenum leakage is due to these components.

It is not possible to know exactly what is the amount of leakage refereed to the edges and what is the amount referred to the floor. It was allocated in this way: 35% of the total leak was assigned to the edges, and 65% was assigned to the floor.

The scope of this work is to analyze the airflow pattern in the airflow plenum, not in the room. For this reason, in the CFD model, the airflow leak through the floor didn't go in the room but it simply disappears. This was possible using the option called "source". Using this option it is possible generate or sink an air flux.

### 5.2.6 Air plenum leakage configuration#2

As described above for this configuration there is a concentrated leakage in a perimeter edge. Anyway the plenum leakage difference between configutation#1 and configuration#2 can not be attributed only to the concentrated leakage. During the plenum set up for configuration#2, the tape that was placed to seal the fissures between the panels, was removed in some points, so this could also be a cause of the higher air plenum leak.

The small area that modeled the concentrated leakage has an area of 0.158 $m^2$ and it is located in the position highlighted in Figure 5-5.

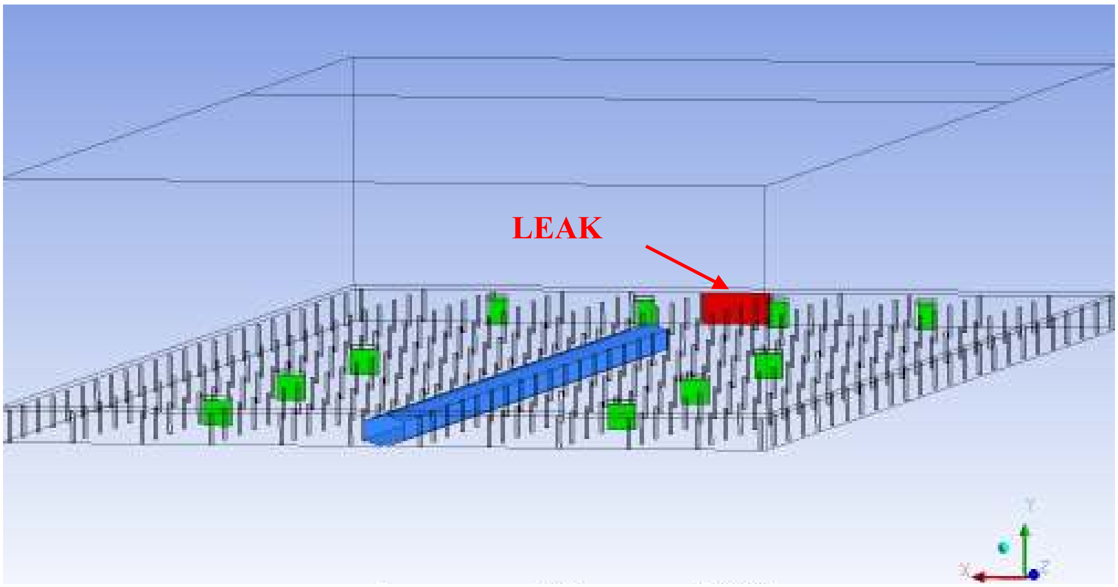



***Figure 5-5 Concentrated leak***

The air plenum leakage difference between the two configuration is 0.05 $m^3/s$, and in the CFD model it was sheared in this way:

- 60 % was assigned to a small area in the plenum edge, where the leakage was located in the real case;
- 40% was assigned to the raised floor.

### 5.2.7 Room CFD model

It was necessary to model the room ambient, even if the interest of this work is focused on the plenum, to better simulate the thermal radiation from the ceiling to the raised floor. For this reason the CFD room model does not have the characteristics of a regular room. Every surface, besides for the ceiling surface, is not a regular wall but it is modeled as an opening. This was done to avoid any control problem for the room temperature, and also because in this way it is possible to better emulate the warehouse conditions.

The ceiling was modeled as a wall with the same temperatures measured during the tests.

In the future, if the attention will be more focused on the airflow pattern inside the room, more attention should be paid for modeling the room ambient and surfaces, but for the scope of this work it should useless.

### 5.2.8 Source term: resistance of pedestals

As Jin et.al discussed in the paper (14), there are two approaches available for representing the resistance of the pedestals in the CFD model:

- actually building the pedestals into the geometry of the CFD model. This approach makes the CFD model complicated from geometry and mesh point of view. The huge mesh size related with the presence of pedestals requires a very powerful computer. This solution is the best choice when air flow pattern inside the plenum is characterized by zones with a strong air velocity and momentum;
- adding a resistance to the fluid domain as a source term in the governing equation to represent the resistance caused by the pedestals. In cases with pedestals uniformly distributed through the plenum, and for a flow pattern not characterized by strong air velocity and momentum, this approach presents many advantages: simplicity, efficiency, and low computational power requirement.

The first approach presented above was used for modeling the plenum configuration#2. The reason for this choice is related with the strong air velocity and momentum for the air plenum inlet. In fact for this configuration the entire airflow enters the plenum from the end part of the duct. The air inlet velocity is around 5 m/s.

For configuration#1 the second approach was chosen. In this configuration the airflow inlets are characterized by a low momentum.

In the CFD plenum model not directly modeling the pedestals (configuration#1), the resistance of those is represented by linear and quadratic resistance coefficients, $C_{R1}$ and $C_{R2}$. An isotropic momentum source can be formulated with the generalized form of Darcy's law (Equation 7):

$$S_M = -C_{R1}U_i - C_{r2}|U|U_i$$

***Equation 7***

where $S_M$ is the momentum source term [kg/m$^2$·s$^2$], and $U_i$ is the velocity [m/s]. This momentum source term is incorporated into Equation 3 to solve the effect of the resistance of pedestals on the airflow velocity in the plenum.

### 5.2.9 Thermal radiation model

To conduct the simulation the Discrete Transfer model was used as thermal radiation model. This model is based on tracing the domain by multiple rays leaving from the bounding surfaces. The technique was developed by Shah (1979) and depends upon the discretization of the equation of transfer along rays. The path along the ray is discretized by using the sections formed from breaking the path at element boundaries. This model is suitable in cases with an optical thickness less than unity across each element. For these simulations " Surface to Surface" transfer mode option was used. This option is used in cases where domain material does not emit, absorb or scatter radiation, like air in the developed CFD domain. This option reduces the computational time without any changes in the results. For a more detailed presentation of the thermal radiation model please refer to (21).

### 5.2.10 Numerical methods

The advection scheme option chosen for the simulations was "High resolution".

The advection term requires the integration point values of $\phi$ to be approximated in terms of the nodal values of $\phi$. The advection schemes implemented in the commercial software used can be cast in the form (Equation 8):

$$\emptyset_{ip} = \emptyset_{up} + \beta\nabla\emptyset \cdot \Delta\vec{r}$$

***Equation 8***

where $\phi_{up}$ is the value at the upwind node, and $\vec{r}$ is the vector from the upwind node to the *ip.* The advection scheme option choice affects $\beta$ and $\nabla\phi$. The "high resolution" scheme uses a special nonlinear recipe for $\beta$ at each node, computed to be as close to 1 as possible. The advective flux is then evaluated using the values of $\beta$ and $\nabla\phi$ from the upwind node. The recipe for $\beta$ is based on the boundedness principles used by Barth and Jesperson (21). This methodology involved first computing a $\phi_{min}$ and $\phi_{max}$ at each node using a stencil involving adjacent nodes (including the node itself). Next, for each integration point around the node, the Equation 8 is solved for $\beta$ to ensure that it does not undershoot $\phi_{min}$ or overshoot $\phi_{max}$. The nodal value for $\beta$ is taken to be the minimum value of all integration point values surrounding the node. The value of $\beta$ is also not permitted to exceed 1(22).

"Auto Timescale" chosen as the fluid time scale control option. This option uses an internally calculated physically time step size based on the specific boundary conditions, and geometry of the domain.

The simulations has been carried out under steady state conditions, but the test facility does not have control of the thermal condition that affects the plenum (such as ceiling temperature and slab temperature), so it was not possible to obtain experimental data under completely steady state conditions. However the thermal conditions in the warehouse building, where the plenum was installed, were not affected by rapid variations but were almost stable, so data collected under these quasi-steady-state (as Hui at.al called them (14)) conditions are acceptable for use in the comparison with theoretical calculations. To further minimize every variation in the thermal behaviors of the plenum every tests were run under constant temperature and airflow inlet for more than two days before collecting the data.

# 6 CFD simulations results

In this chapter the simulation results for the two models are presented. The attention is focused on the parameters that are suitable for the comparison with the experimental data.

## 6.1 Airflow pattern: configuration#1

The best way to represent the airflow pattern predicted by the CFD program is using images in RGB colors that represent the air velocity in a plenum section plane, and using the stream-lines.

In Figure 6-1 a plane, located at the middle height of the plenum and colored as a function of the air velocity, is presented.

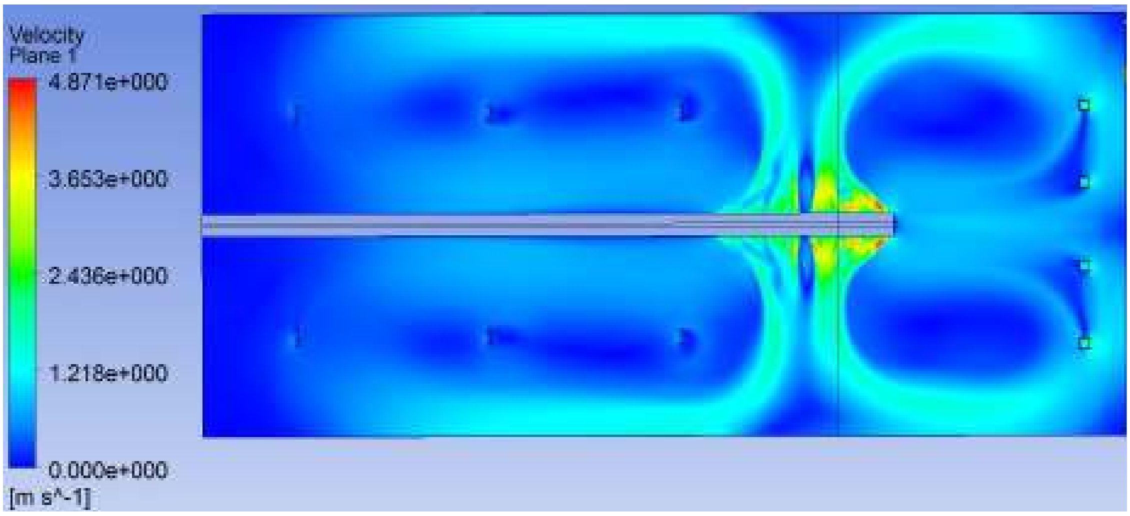



***Figure 6-1 Air velocity field. Configuration#1***

As it can be seen in Figure 6-1, there are four big swirls in the plenum, two located in the perimeter, and two located in the interior part. The air that comes out from the holes is split in two parts, one goes in the perimeter and the other one goes in the interior. As discussed previously, during the tests the sum of airflow through the perimeter diffusers was the same as the sum of airflow through the interior diffusers, so the phenomena just mentioned are easy to explain.

In Figure 6-2 the stream lines are used to visualize airflow pattern inside the plenum.

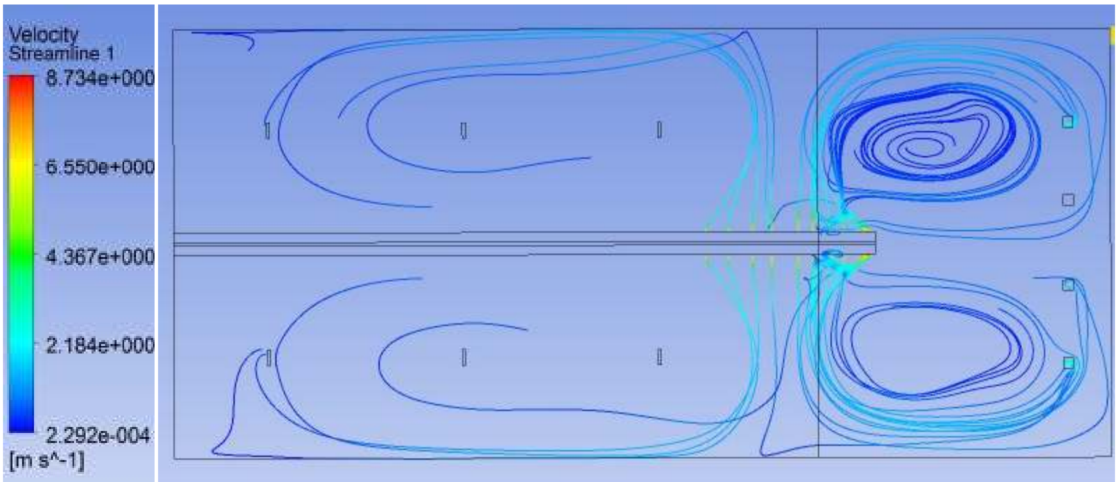



***Figure 6-2Airflow pattern, stream lines representation***

## 6.2 Airflow pattern: configuration#2

In Figure 6-3 the air velocity field for the configuration#2 is presented. In the image the presence of pedestals is clearly visible.

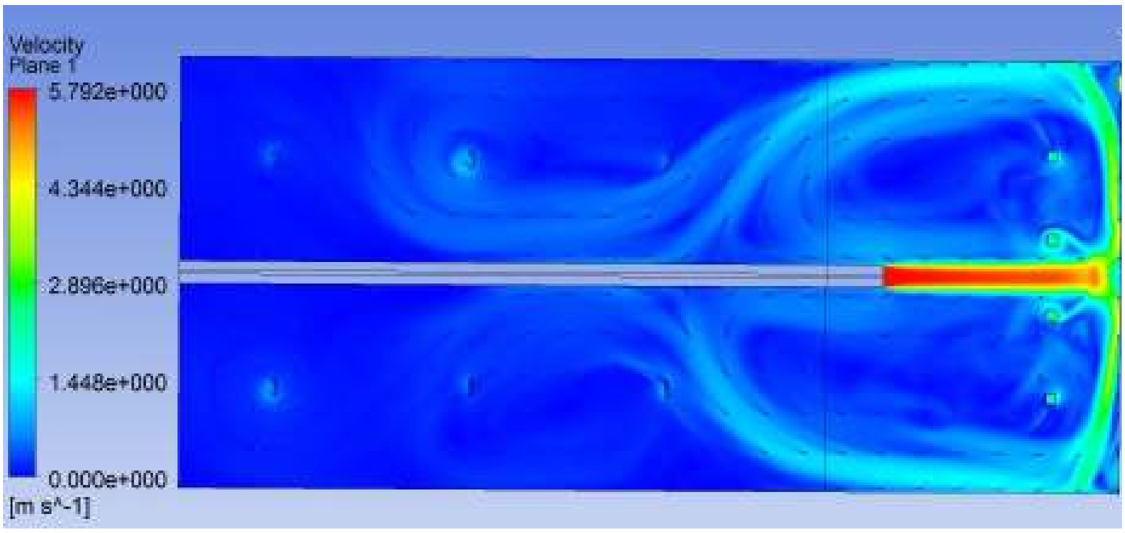



***Figure 6-3 Air velocity field. Configuration#2***

It is not easy to identify the airflow pattern in Figure 6-3, and the stream lines can help to clarify the air pattern configuration (see Figure 6-4).

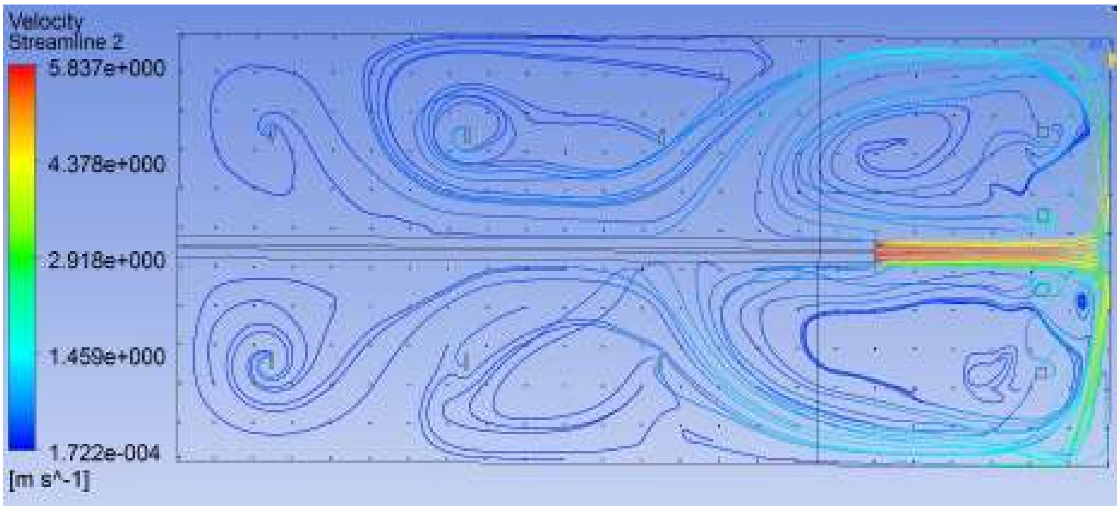



***Figure 6-4 Airflow pattern, stream lines representation***

Blanding the two figures can give a nice representation of the velocity field (see Figure 6-5).

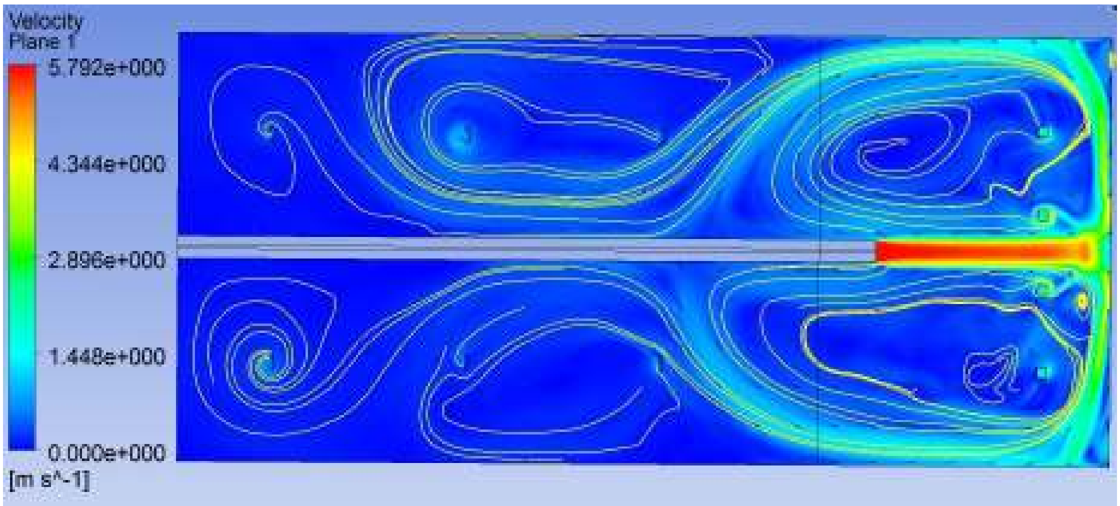



***Figure 6-5 Airflow pattern. Configuration#2***

## 6.3 Temperature field: configuration#1 and #2

The temperature field for configuration#1 (Figure 6-6), and for configuration#2 (Figure 6-7), can be visualized using the same plane presented above for the velocity field.

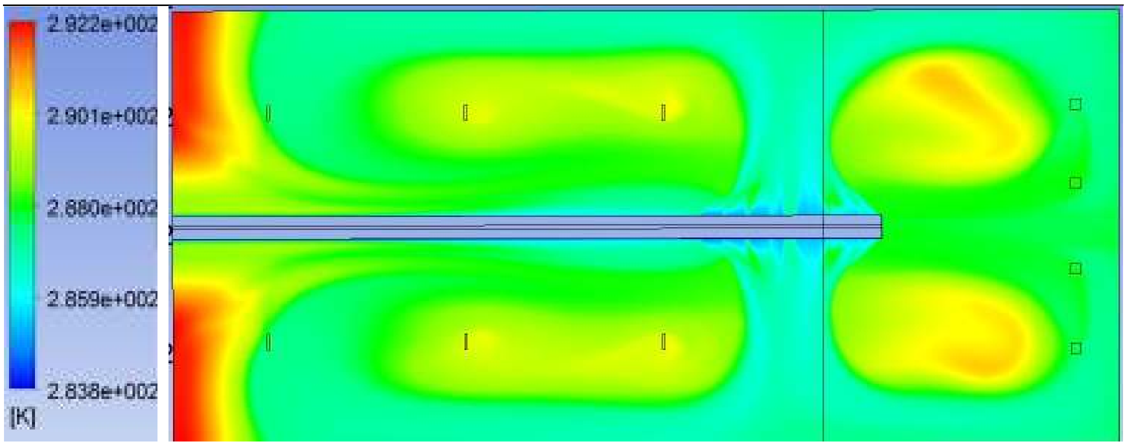



*Figure 6-6 Plenum temperature field. Configuration#1*

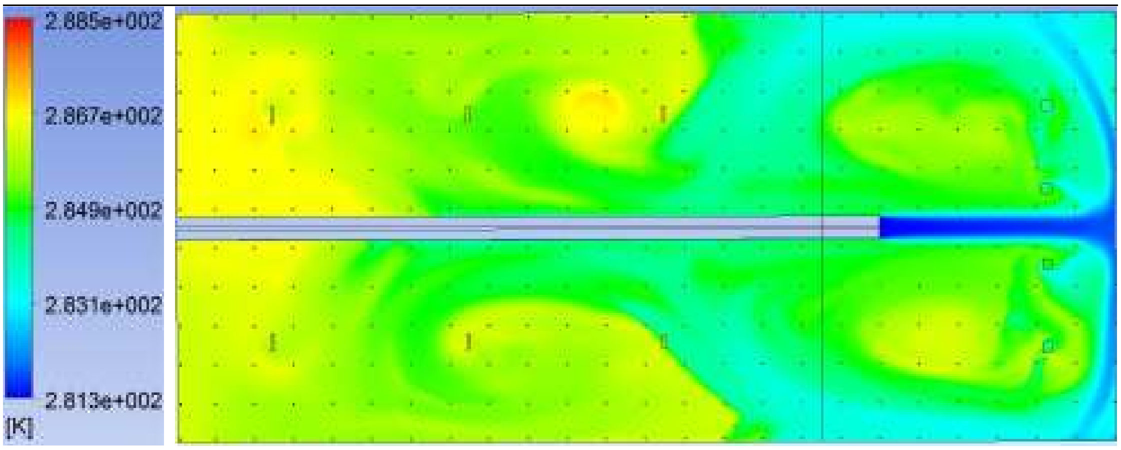



*Figure 6-7 Plenum temperature field. Configuration#2*

## 6.4 Diffusers airflows and temperatures

Using the Computational Fluid Dynamic program it was possible to estimate air flows and air temperatures through every diffuser. The values are depicted in Figure 6-8 and Figure 6-9 for configuration#1, and in Figure 6-10-Figure 6-11, for configuration#2.

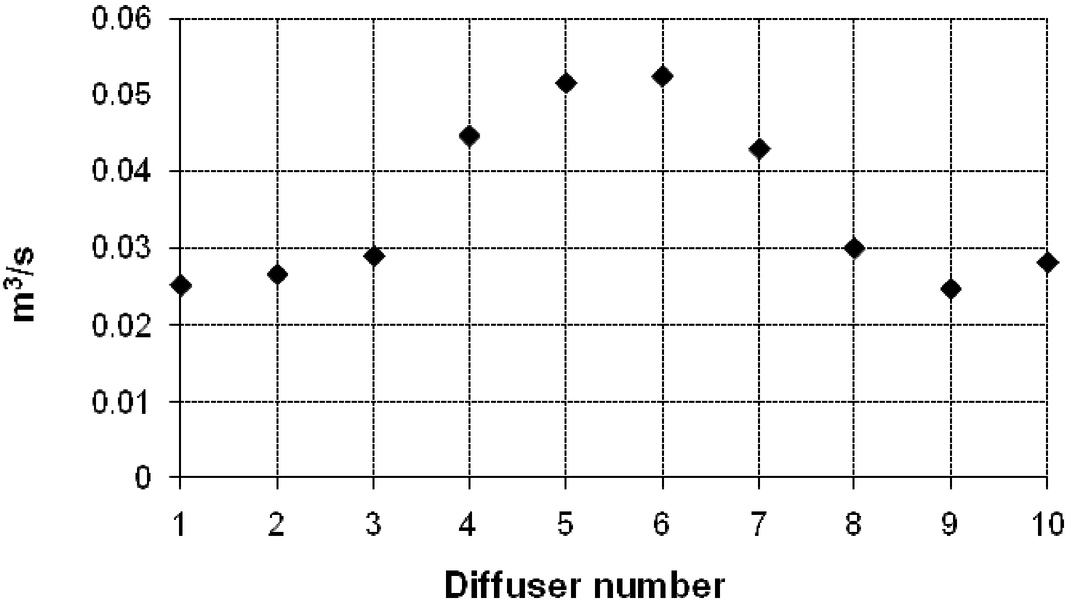



*Figure 6-8 Diffusers airflows. Configuration#1*

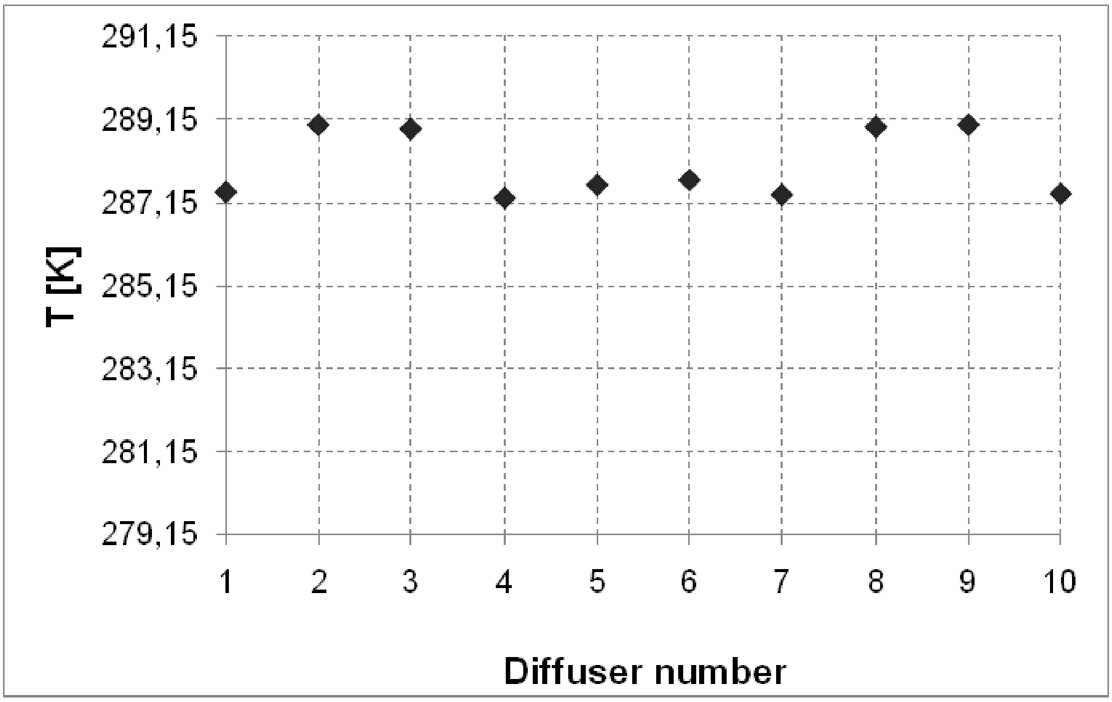



*Figure 6-9 Diffusers temperatures. Configuration#1*

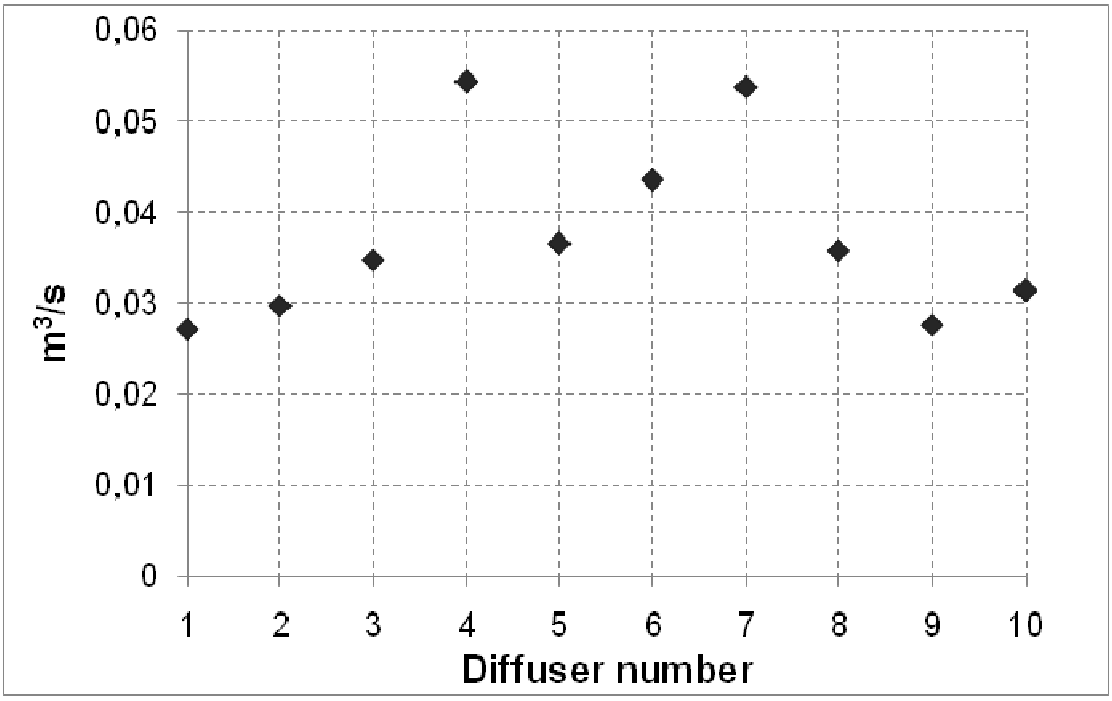



*Figure 6-10 Diffusers airflows. Configuration#2*

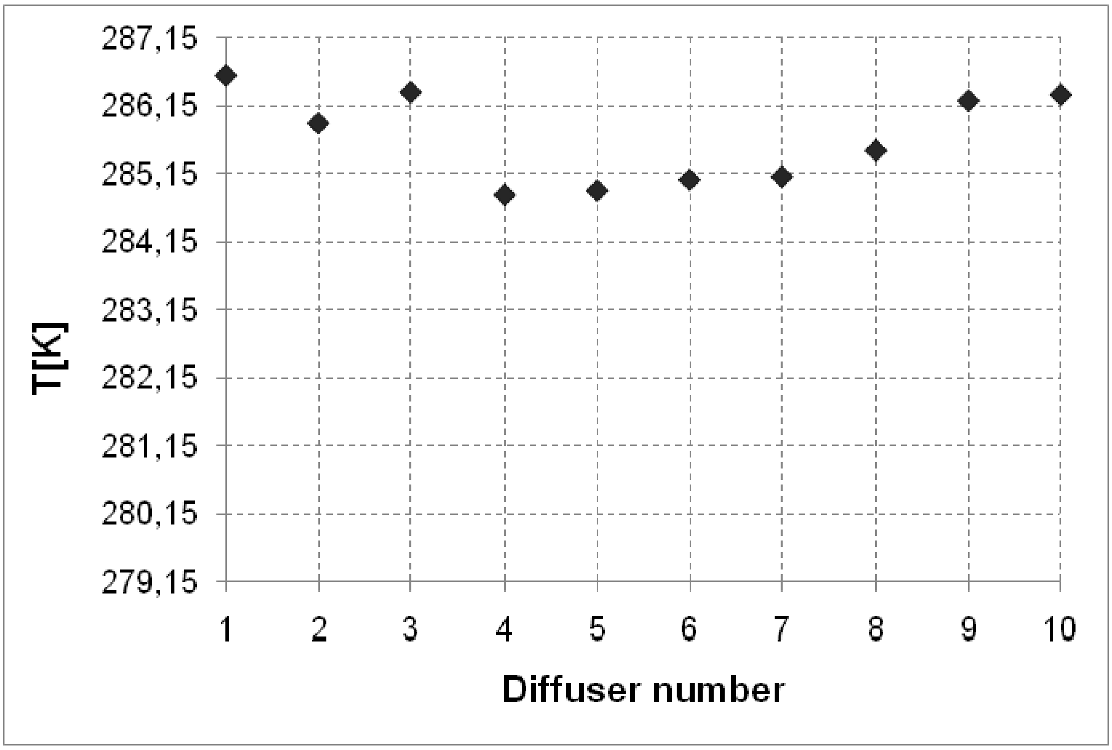



*Figure 6-11 Diffusers temperatures. Configuration#2*

# 7 CFD models validations

The validation studies must demonstrate that the accuracy of predictions is sufficient to meet the stated objective of the model. There is a need for well-defined data, from laboratory experiments and full-scale tests, for model validation.

The well-defined data used to validate the CFD models are the data collected during the experimental tests with the plenum facility .

The models were developed based on the plenum facility's main dimensions, and using as boundary conditions the same values as those measured during the tests, so if the CFD model works properly, the results should be almost equal to the experimental measurements.

The validation work was focused on the diffusers parameters: airflow through the diffuser, and air temperature. These parameters were chosen because they can be considered as the "interface parameters" between the underfloor plenum and the room ambient. If the model is able to predict the airflow and the air temperature for every diffuser, it means that the model has correctly simulated the airflow pattern and the heat exchange inside and through the plenum.

In order to evaluate the goodness of the results a robust scientific method must be used. For this work the method presented by Zhang Z. et al. (18) was selected, which was used by the authors to evaluate the goodness of some CFD predictions of air temperature distribution in a conditioned room. This method is composed of four letters to classify the accuracy of the prediction: A,B,C,D. The method uses the relative error between prediction and measurement at measured points as a major criterion. If this error is less than 10% or larger than 50% at most measured points, the model accuracy is rated as A or D, respectively. While ratings A and D quantify the extremes, the difference between B and C can be more subtle. Rating B is given to predictions with relative error less than 30%. Rating C is given to the remaining predictions. The relative error calculations for the temperature were based on the nominal temperature differential (18). In our case the nominal temperature differential is the difference between the plenum inlet temperature and the average, interior or perimeter, outlet temperature, measured respectively for the perimeter diffusers or for the interior diffusers. It is possible to assign a global rate to the model based on all the rates given to the diffusers.

## 7.1 CFD model validation, Configuration#1

In this chapter the validation for the CFD model, based on configuration#1, is presented. The measured values and predicted values are presented in the following charts, and to evaluate the errors entity the method presented by Zhang at al. was applied.

In Figure 7-1 and Figure 7-2 the differences, respectively in terms of temperature and airflow, between the model predictions and the measured data are shown.

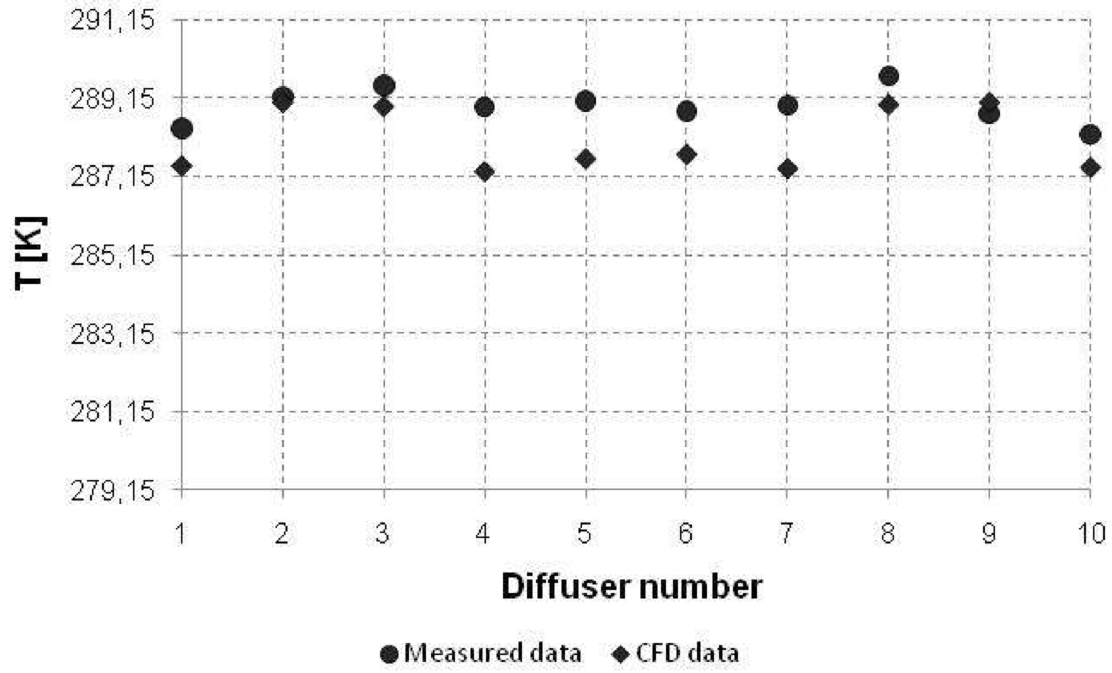



*Figure 7-1 Diffusers air temperatures comparison, configuration#1*

As can be seen in Figure 7-1, predicted air temperatures are always cooler than measured data. This could be caused by a wrong estimation of the slab thermal resistance. In fact, due to the location of plenum facility in a preexistent warehouse, it was not possible to estimate the real value of thermal resistance. Anyway the model is able to qualitatively predict the air temperature trend. This entails a good level of prediction for the airflow pattern, and heat exchange phenomena inside of the underfloor plenum.

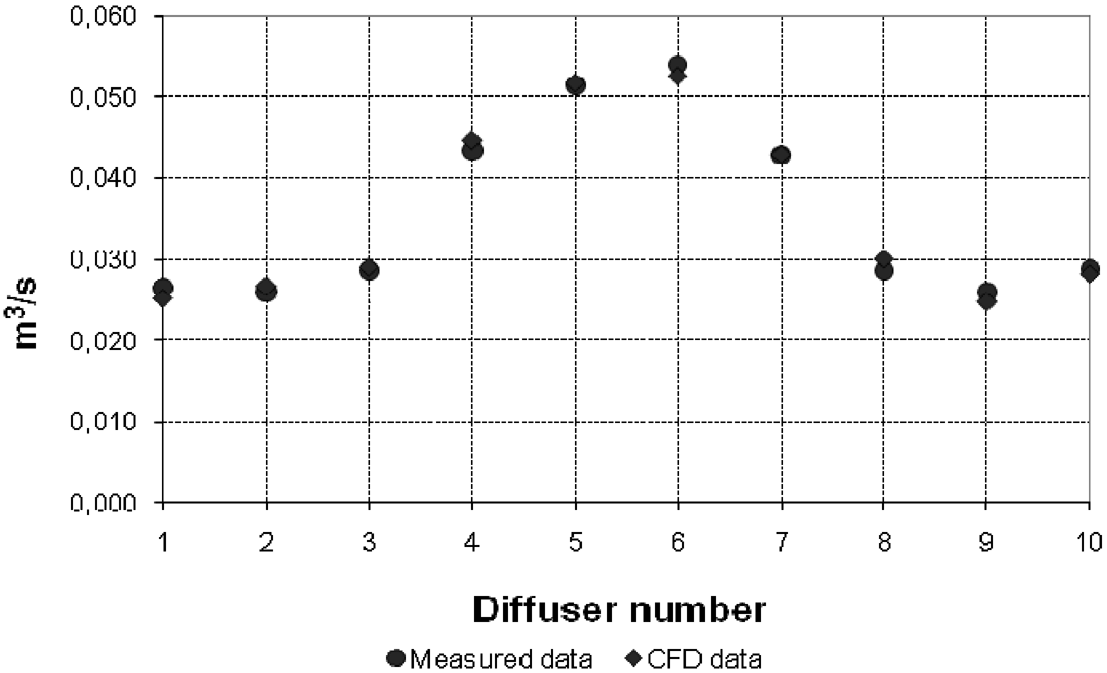



*Figure 7-2 Diffusers airflows comparison, configuration#1*

A very good agreement for the airflows is evidenced in Figure 7-2.

Using the evaluating method presented above it was possible to obtain the values presented in Table 6 and Table 7, in which the agreement between CFD results and measured values are quantified respectively in terms of airflow and air temperature.

*Table 6 Diffusers airflow ratings, configuration#1*

| Diffuser number | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| Rating | A | A | A | A | A | A | A | A | A | A |

*Table 7 Diffusers air temperature ratings, configuration#1*

| Diffuser number | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| Rating | B | B | A | C | B | B | C | B | A | B |

As mentioned above the agreement for the CFD diffusers airflows with the measured values is very good. The error is less than 10% for every estimated airflow. For the CFD predicted temperatures the agreement with the experimental data is not very good like the airflow results. The values are still good but there are two C that highlight some inaccuracies. The most probable cause is the wrong estimation of the slab thermal resistance.

The CFD model, which reproduces the configuration#1, has shown very good capabilities to predict the plenum airflow pathway, and rather good capabilities to simulate the heat exchange phenomena into the plenum. The model capability to estimate the plenum airflow pathways can be globally rated “A”, while its capability to simulate the heat exchange phenomena can be rated “B”.

The overall evaluation of the model is good. This CFD model can be used to estimate the performances of underfloor plenums using fabric, or metal, ducts to improve the temperature distributions.

## 7.2 CFD model validation, Configuration#2

This model presented some more difficulties compared with the other model. First of all the concentrated leakage introduced an ulterior complexity into the model, and more then this it introduced a strong approximation. In fact it was not possible to construct some specific tests to estimate all the features of the leakage, such as the amount airflow leaking through the concentrated leakage, but those were deduced analyzing the data. The most evident effect of the concentrated leakage is the unusual cold temperature measured for the diffuser number eight, and similarly unusual warm temperature measured for the diffuser number seven (the diffusers numeration is the same presented in Figure 4-17 and Figure 4-20). The CFD model has shown some difficulties to reproduce this phenomena. Another complexity was introduced into the model by the pedestals modeling. This required a quite high number of cells for the mesh, with a consequent increase in the simulation time.

The charts with the comparisons between the measured data and the predicted data for the diffusers airflows and air temperatures are reported in Figure 7-3 and Figure 7-4.

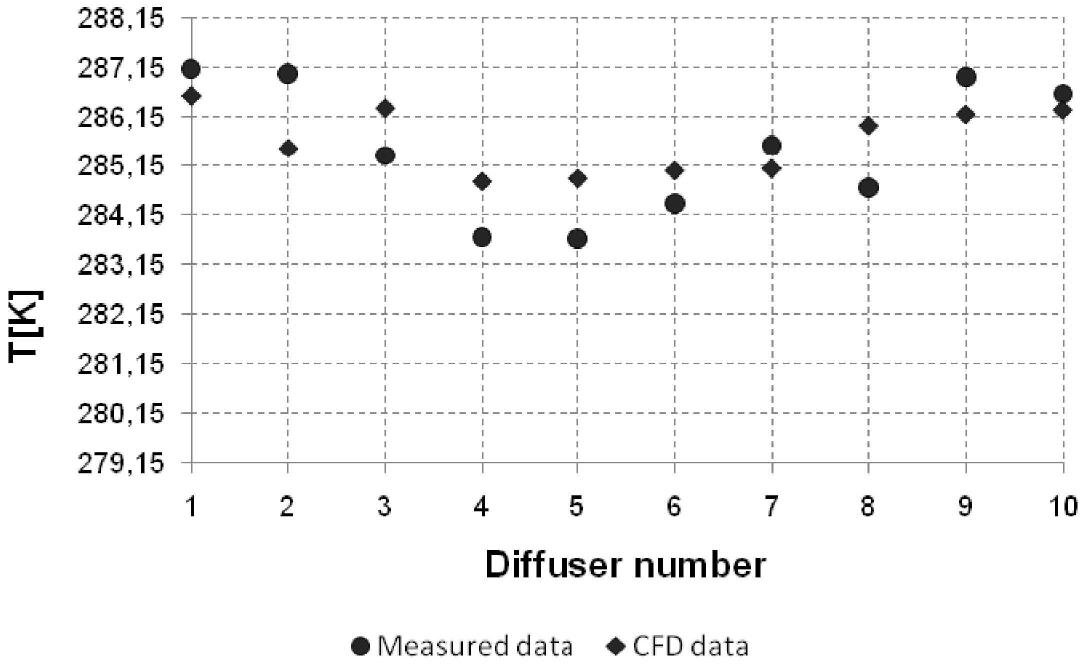



*Figure 7-3Diffusers air temperatures comparison, configuration#2*

The CFD prediction of diffuser air temperatures shown in Figure 7-3 is more linear than for the real case, and the temperature anomaly measured for diffusers number seven and eight is not fully predicted by the model.

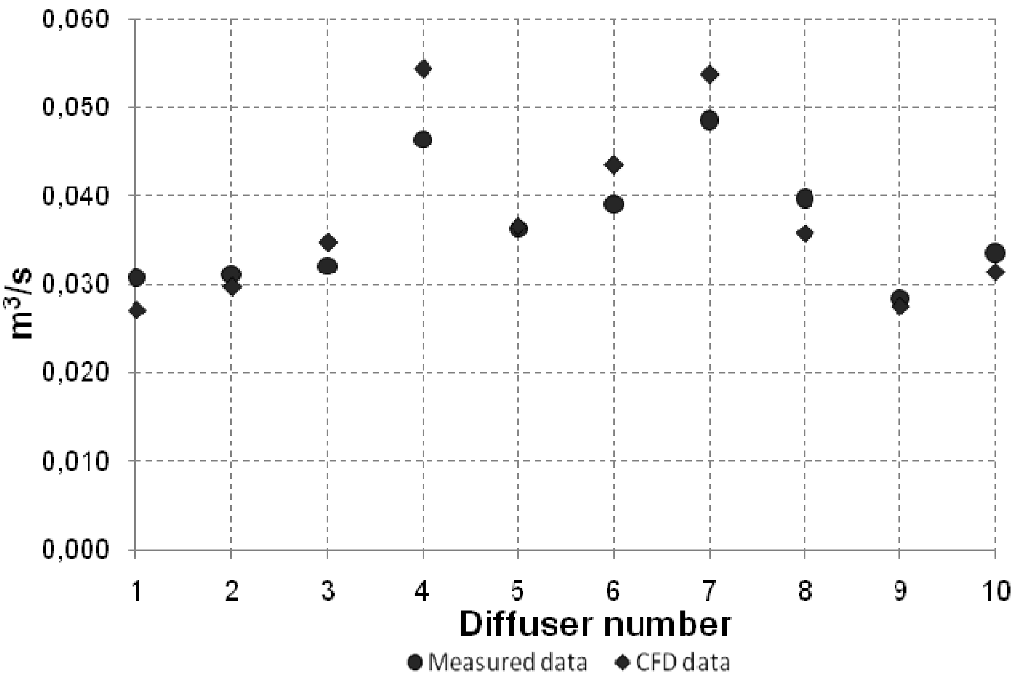



*Figure 7-4 Diffusers airflows comparison, configuration#2*

Similar to the configuration#1 case this model also shows a good level of agreement in terms of airflows predictions.

A better explanation of the model quality can be given using the evaluation method. Table 8 and Table 9 respectively report the ratings for the agreement in terms of airflows, and air temperatures.

***Table 8 Diffusers airflow ratings, configuration#2***

| Diffuser number | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| Rating | B | A | A | B | A | B | A | A | A | A |

***Table 9 Diffusers air temperature ratings, configuration#2***

| Diffuser number | 1 | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---|---|---|---|---|---|---|---|---|---|---|
| Rating | A | C | B | C | C | B | B | B | B | A |

In the previous analysis, based only on the Figure 7-3, the attention was focused on diffusers seven and eight, while in Table 9 no such problem can noticed with these two diffusers, while the diffusers number two, four and five bore evidence of some problem. Also in this case the cause of these problems can be related with the location, dimension, and modeling of concentrated air leakage, but this analysis can give one more information: the error strongly affects the one side of the plenum compared with the other. This phenomena can be related only with the concentrated leakage because it is the only element not symmetric in the model. Every element in the model (diffusers, box diffusers etc.) has an identical element in the symmetric side of the plenum with respect to the duct axis, except for the concentrated leakage. This reinforces the hypothesis that the leakage and its modeling can strongly influence the plenum airflow pathways. In this case the attention is focused on some pathways problem unlike previous model where the attention was focused on some heat exchange problem. The reason behind that can be explained by two points:

- for the previous model Table 7 shows a systematic error for every temperature prediction. So if it was the problem with the airflow pathways, it would evidence some positive and some negative differences between the measured and the predicted data;

- the heat power exchanged by air through the plenum can give a strong indication of the phenomena. Using Equation 9 it is possible to estimate the heating power exchanged by the plenum for the CFD model and during the test, for the configuration#2;

$$P = \sum_{i=1}^{n} \dot{m}_i \, c_p (T_{in} - T_i)$$

***Equation 9***

where i is the diffuser number, it goes from 1 to 10; $\dot{m}_i$ is the airflow through the diffuser i; cp is the air specific heat capacity; Tin is the plenum air inlet temperature and Ti is the air temperature through the diffuser i. Using the Equation 9 the plenum heating power exchanged turns out to be 1758 W for the real case, and 1890 W for the CFD model. The difference between the two values is very small, less than 8%, which means that the CFD model can predict the heating exchange phenomena in the plenum with a very good level of agreement with the experimental data.

From the above evaluation it may be concluded that the model shows a good capability to simulate the heat exchange phenomena in the plenum but has some small problems in predicting the plenum airflow pathways. Anyway the overall performance of this model is acceptable.

## 7.3 Considerations on CFD models

Both CFD models have shown good capabilities of predicting the heating exchange and airflow phenomena in the underfloor plenum. The objective of the modeling work was achieved. The model made for a fairly simple plenum configuration can be extended and applied to other more realistic plenum configurations.

The test results, presented in the chapter 4, have shown that using flexible duct in underfloor plenum can reduce the perimeter airflow temperature. So this finding supports CBE's recommendation of reducing the perimeter airflow temperature. Now the question is, how does this solution work compared with the other commonly used solutions? A common solution to reduce the air temperature in the perimeter consists in using plenum inlets with higher inlet velocities directed at critical perimeter locations. The next part of this work is focused on the comparison between this last solution and the flexible duct solution, using a CFD model based on the two validated configurations in this chapter but with more realistic dimensions and configurations.

# 8 New CFD model configuration

The new CFD configuration is based on the CFD model developed and validated at the Center for the Built Environment (14). The researchers have conducted CFD simulations, using CBE's validated underfloor plenum model, to investigate the temperature distributions in an underfloor plenum for different ways (number of inlets and inlet momentum) supply air into the plenum. A part of their work is reported below.

## 8.1 Original model configurations

The term "original" in the title is referred to the configurations used by the CBE's researchers during their tests, which are different compared the configurations used in this work, so all the information reported in this sub-chapter comes from the work did by Phan et al. at CBE.

The validated CFD model was used to simulate different plenum configurations to investigate the impact of inlet velocity, airflow rate, inlet jet momentum and inlet jet direction on the interior zone vs. perimeter zone temperature distribution. Schematic diagram of one of these different plenum configurations is shown in Figure 8-1. The modeled plenum has dimensions of 15.24 m by 30.48 m by 0.3 m high. Air supply inlets are located at the long edge of the plenum. Hence, the plenum dimension of 15.24 m in the direction of air inlet represents the maximum practical distance between the plenum inlet and the point of discharge into the space. In order to follow design practice closely, supply air is assumed to be delivered into the plenum from the interior zone and to travel a certain distance before serving the outlets in the perimeter zone. As Figure 8-1 shows, the perimeter zone was assumed to be 4.54 m in depth (a common practice for energy simulation) and the interior zone is whatever is 10.7 m long. For these simulations, there were 40 diffusers in total, 10 in the perimeter zone and 30 in the interior zone, plus two fan coil units. Figure 8-2 shows schematic diagram of the boundary conditions established for the temperature distribution analysis cases. These primary boundary conditions were as follows: inlet air temperature is 16.7°C, ceiling emissivity is 0.9 and view factor is 0.5. Ceiling temperature was set to 25.6°C. The total airflow rate was set to

2.206 m3/s for most cases, with an average diffuser airflow rate of 0.032 m3/s, an average interior airflow rate of 0.945 m3/s, and an average perimeter airflow rate of 1.261 m3/s.

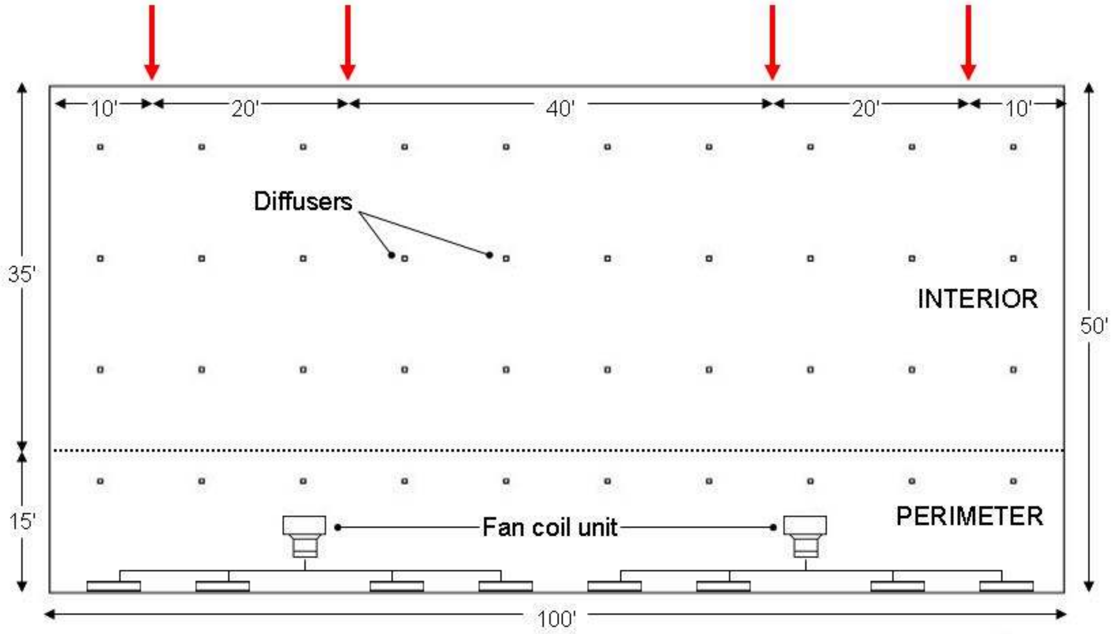
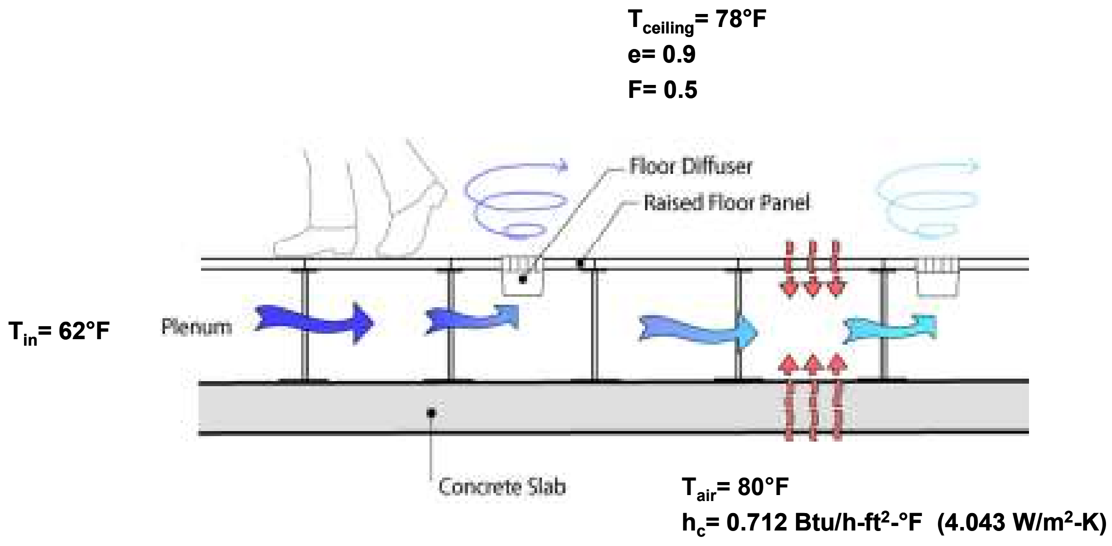



*Figure 8-1 One of the original configurations of CFD model (the unit dimension for the measures is ft)*




Floor conductivity = 1.36 Btu-in/h-ft$^2$-°F ( 0.196 W/m-K)
Floor thickness = 1.3 inch
Slab conductivity = 6.46 Btu-in/h-ft$^2$-°F (0.93 W/m-K)
Slab thickness = 6 inch

*Figure 8-2 Original CFD boundary conditions*

## 8.2 New CFD model configurations, combination with EnergyPlus

The CFD model used by the CBE's researcher, has a realistic configuration both in terms of dimension and diffuser number. It could be a floor of an open-plan office, representing a part of the building. Also the boundary conditions are quite realistic, but those are not based on real data, but instead are based on simulations.

In order to improve the connection with a real case, it was used with an energy simulation program (EnergyPlus (24)), to get the boundary conditions for the CFD model.

The floor model was considered to be a part of a bigger floor located at the second floor of a three storey office building. Three types of interior cooling loads (people, equipment and lighting) were simulated, and for all of these a schedule was created. The EnergyPlus model implemented the underfloor plenum model developed in the Center for the Built Environment (24), (12). The energy simulations were run for a summer design day, and the weather data of Sacramento, California were used.

During the daily operation of the HVAC system, the cooling load is not constant, but it varies from 100% of peak load to 0%. This strong variation could influence the airflow pathway inside the underfloor plenum. To analyze this phenomenon four cooling load conditions were selected to be simulated with the CFD model: 25%, 50%, 75% and 100% of peak load.

The CFD simulations were done under steady-state conditions but in this way the model does not take into account of the slab thermal mass, that represent an important element in the energy simulations. To avoid this problem the temperatures, evaluated with EnergyPlus, of the bottom surface of the raised floor, and of the top surface of the concrete slab, were used as boundary conditions for the CFD model. Using these temperatures as boundary conditions it was no longer necessary to simulate the conditioned space above the plenum, and the thermal radiation.

From here on the term ceiling is referred to the bottom surface of raised floor, and the term slab it is referred to the top surface of concrete slab.

Figure 8-3 shows a schematic representation of the plenum model, and in Figure 8-4 the CFD model is presented.

One conclusion of this research is that the average perimeter zone temperature is always higher than the average interior zone temperature when the number of inlets is six or greater. Unfortunately, increasing the inlet velocity does not help to deliver cooler air into the perimeter zone. The perimeter zone temperature increases slightly instead.

One of the lowest perimeter temperature values was obtained using only four plenum inlet. This configuration, presented in Figure 8-3, was used for this work.

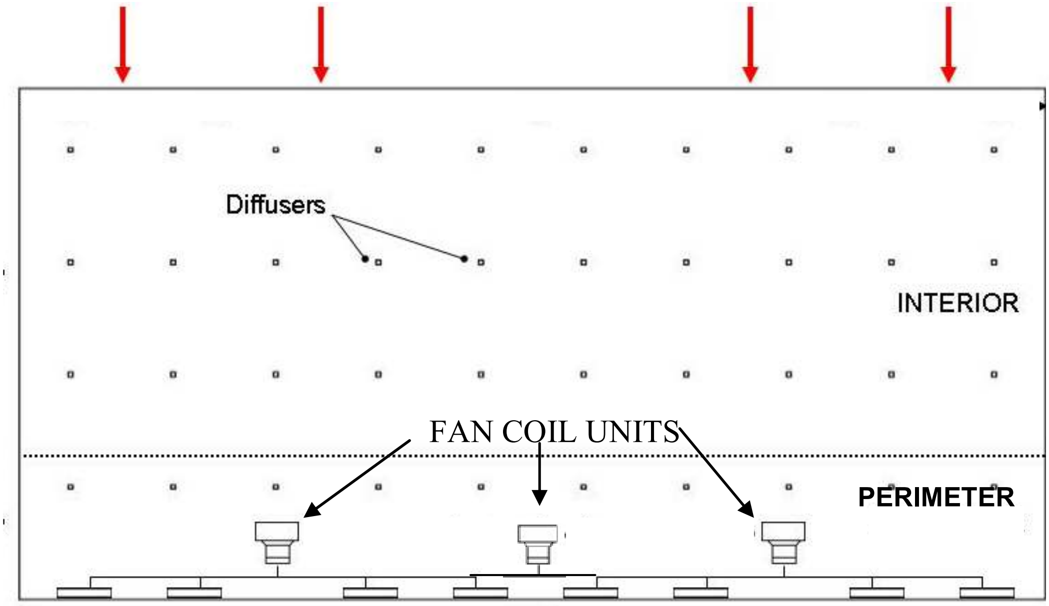



*Figure 8-3 Schematic representation of the plenum model*

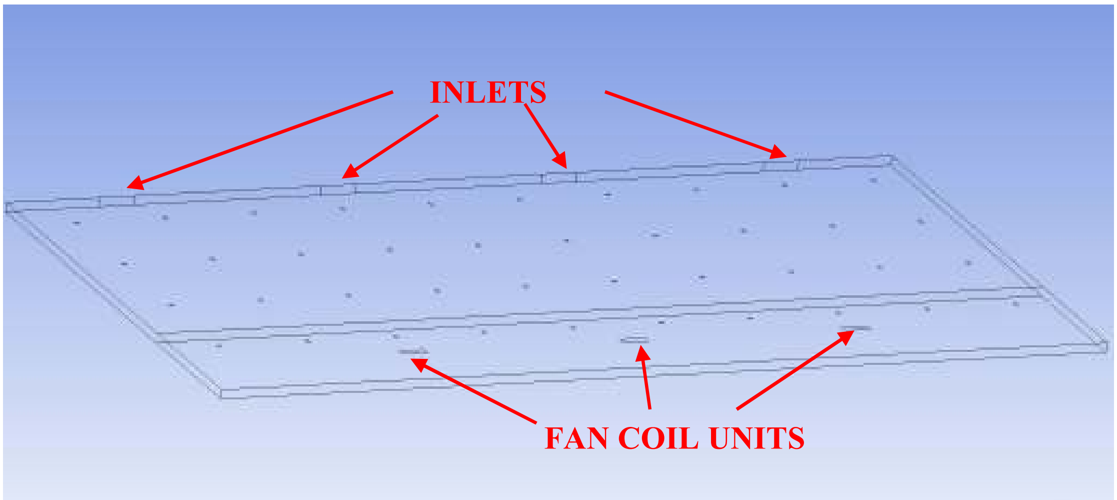



*Figure 8-4 CFD model*

The boundary conditions for the CFD model, which are deduced from energy simulations, are reported in Table 10 and Table 11 for the four load conditions. The boundary conditions are divided in interior and perimeter, with the meaning showed in

Figure 8-3. In Figure 8-4 it can be seen that the ceiling and the slab are divided in two parts, the perimeter part and the interior part. Also the airflow it is divided in interior and perimeter. This is because the energy simulation program can estimate the airflow demand for the interior and for the perimeter. In the CFD model the plenum airflow for each of the four loads is the sum of the interior and perimeter airflows. The perimeter airflow is always higher than the interior airflow, this is due to the higher cooling load for the perimeter part of the above conditioned room (i.e.. solar radiation).

***Table 10 Perimeter zone CFD boundary conditions***

| Perimeter conditions | | | |
|---|---|---|---|
| Load condition | Ceiling temperature [°C] | Slab temperature [°C] | Airflow [$m^3$/s] |
| 25 % (8/9 8am) | 22.6 | 22.5 | 0.539 |
| 50 % (10 pm) | 23.0 | 22.8 | 1.077 |
| 75 % (8 pm) | 22.9 | 22.7 | 1.548 |
| 100 % (2 pm) | 22.9 | 22.7 | 2.222 |

***Table 11 Interior zone CFD boundary conditions***

| Interior conditions | | | |
|---|---|---|---|
| Load condition | Ceiling temperature [°C] | Slab temperature [°C] | Airflow [$m^3$/s] |
| 25 % (8/9 8am) | 22.3 | 21.9 | 0.452 |
| 50 % (10 pm) | 22.1 | 21.8 | 0.499 |
| 75 % (8 pm) | 21.8 | 21.6 | 0.452 |
| 100 % (2 pm) | 21.9 | 21.9 | 0.514 |

Since in the perimeter there are only ten small round (swirl) diffusers compared with the thirty in the interior part of the plenum, and since the perimeter airflow is higher compared to the interior airflow, the biggest part of the perimeter airflow was assigned to the fan coil units, which is typical of realistic systems.

Each fan coil unit was modeled as an "outlet". which represented the inlet to the fan coil unit. As shown in Figure 8-3, the fan-coil units are ducted to linear floor grilles, so the airflow does not participate in the plenum air distribution once it enters the fan-coil unit. Each fan-coil "outlet" extracts from the plenum a pre-defined airflow that was defined using the following equation:

$$\dot{m} = \left[\frac{\dot{m}_{perim} - (\frac{\dot{m}_{int}}{30} \cdot 10)}{3}\right]$$

***Equation 10***

where $\dot{m}$ is the airflow through the every fan coil unit; $\dot{m}_{perim}$ is the perimeter airflow; $\dot{m}_{int}$ is the interior airflow.

The plenum inlet air temperature for all simulations was fixed to 16.7°C.

## 8.3 CFD model of the plenum equipped with flexible ducts

The scope of this work is to compare a regular plenum system, which uses the high air inlet momentum to deliver fresh air in the perimeter, with a system that uses flexible duct to improve the air distribution. The CFD model presented in Figure 8-3 and Figure 8-4 represents the first system. For the configuration with the flexible ducts a new model was built. The new model has the same dimension and boundary conditions of the open wide plenum, with the addition of four flexible duct models. For this new configuration "DuctSox" company was contacted, it is one of the producers of fabric ducts for ventilation system. With the help of an expert of this company it was possible to design two flexible ductwork configurations for the underfloor plenum, under these operative conditions. The two configurations are presented in Figure 8-5 and **Errore. L'origine riferimento non è stata trovata.**.

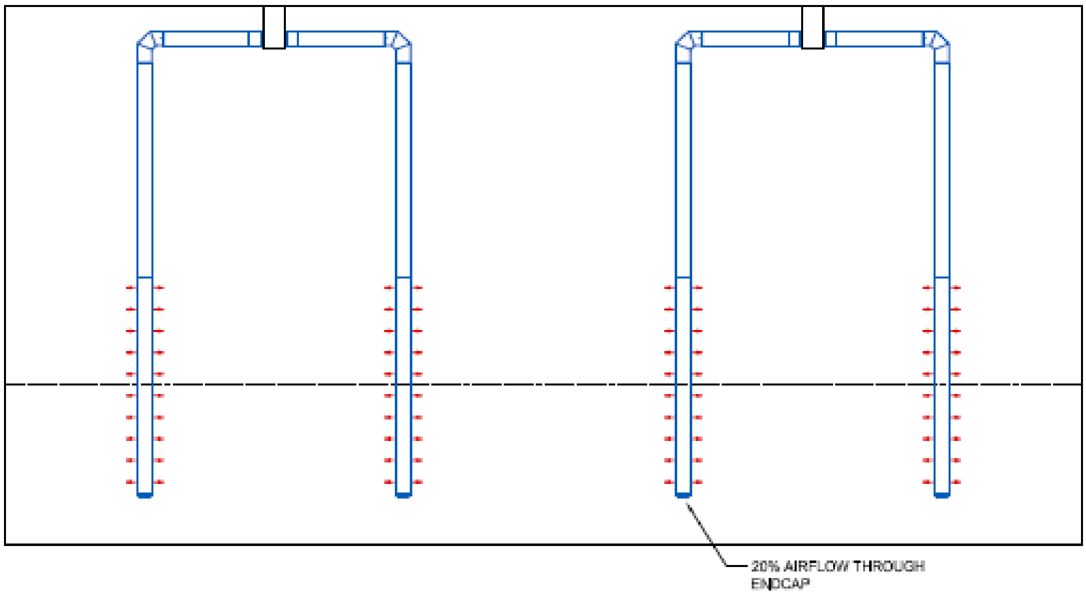



*Figure 8-5 Fabric ductwork, design#1*

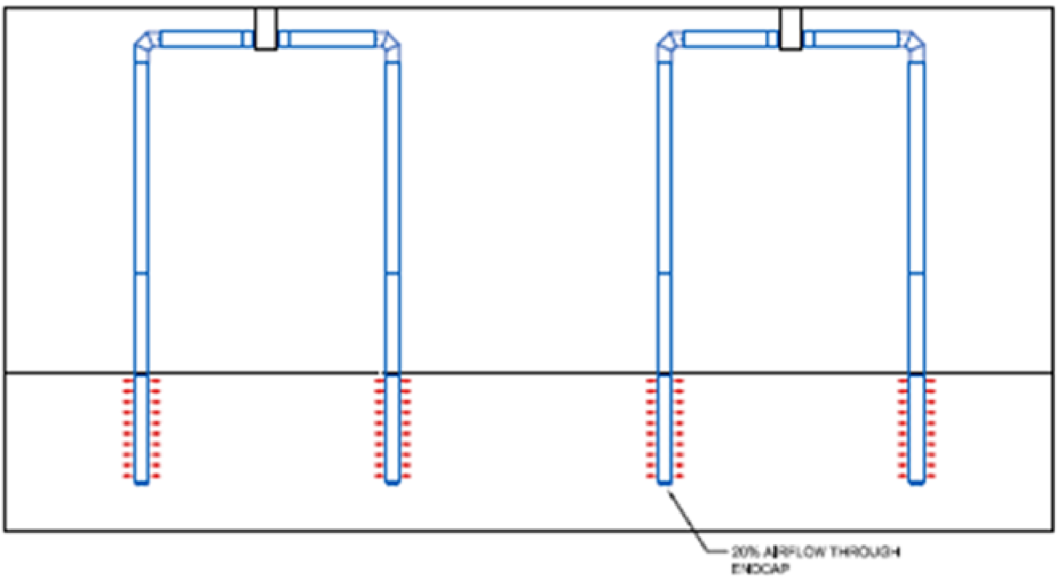



*Figure 8-6 Fabric ductwork, design#2*

The difference between the two ductwork configurations is in the end-caps:

- in the configuration#1 the 20% of total airflow enters in the plenum through the four operable end-caps (see Figure 8-7). The opening in the end-cap is a seven inch diameter hole. The rest of air enters into the plenum through the six vented meters of every duct;

- in the configuration#2 the four end-caps are the same, but the vented parts of ducts are three meters long.

Between the two configurations, configuration#1 was chosen, because it works better for this plenum configuration.

The CFD model of this configuration is shown in Figure 8-8.

The pressures inside the fabric ducts, and the relative air leakage through the fabric, were estimated using a spreadsheet that is used by the designer to estimate the ducts operative conditions.

All the boundary conditions related with the ducts, for the four load conditions, are reported in Table 1.

***Figure 8-7 Operable end cap***

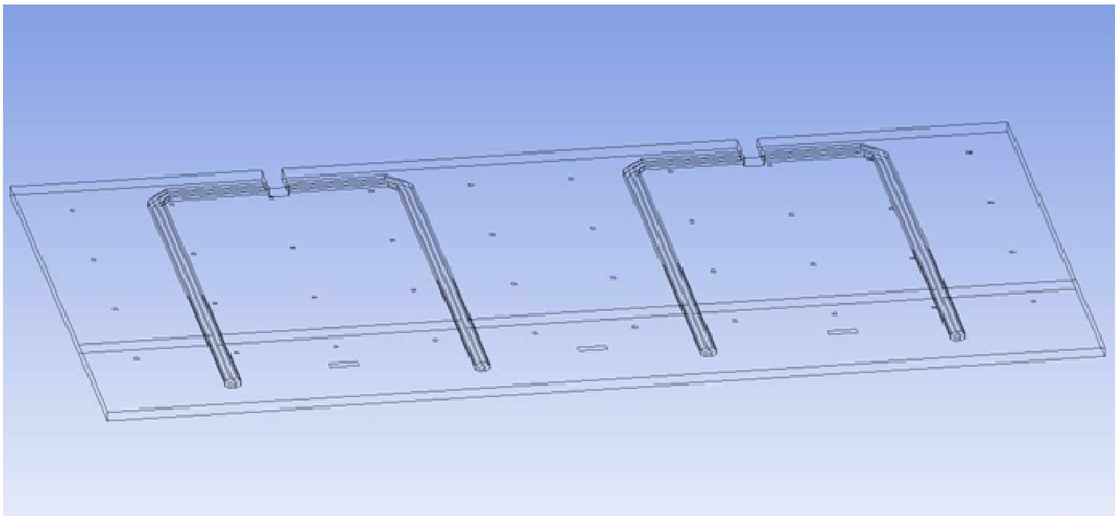


*Figure 8-8 CFD model of the ductwork configuration*

***Table 12 Fabric ducts boundary conditions***

| Fabric ducts boundary conditions | | | |
|---|---|---|---|
| Load condition | Airflow through the vent part [$m^3/s$] | Airflow through the end caps [$m^3/s$] | Air leak through the fabric [$m^3/s$] |
| 25 % (8/9 8am) | 0.688 | 0.276 | 0.026 |
| 50 % (10 pm) | 1.144 | 0.391 | 0.042 |
| 75 % (8 pm) | 1.537 | 0.410 | 0.053 |
| 100 % (2 pm) | 2.108 | 0.559 | 0.068 |

The pedestals were not directly simulated, the Equation 7 was used to take into account pedestals resistance.

For all the set up configurations please refer to chapter 5.2.10.

# 9 CFD simulation results

In this chapter the CFD results for the no ducted plenum model, henceforth referred to as configuration#1, and the CFD results for the ducted plenum model, henceforth referred to as configuration#2, are presented.

For every load two different results are presented: 1)temperature field for a section plane located at the middle height of the plenum; 2)velocity field for the same section plane.

## 9.1 Configuration#1, temperature and velocity fields

In the figures from Figure 9-1 to Figure 9-4 the temperature fields for the four load conditions for the configuration#1 are depicted in RGB colors.

As it can be seen from the figures, especially from Figure 9-1, as the plenum inlet airflow is reduced less cool air reaches the plenum perimeter. For the case that simulates 25% of peak load, only a small amount of cool air can reach the plenum perimeter. If at the peak cooling load the air inlet momentum is strong enough, this is not necessarily true for all the partial load range, and it must kept in mind that in real cases for the main part of time the HVAC system works with partial loads.

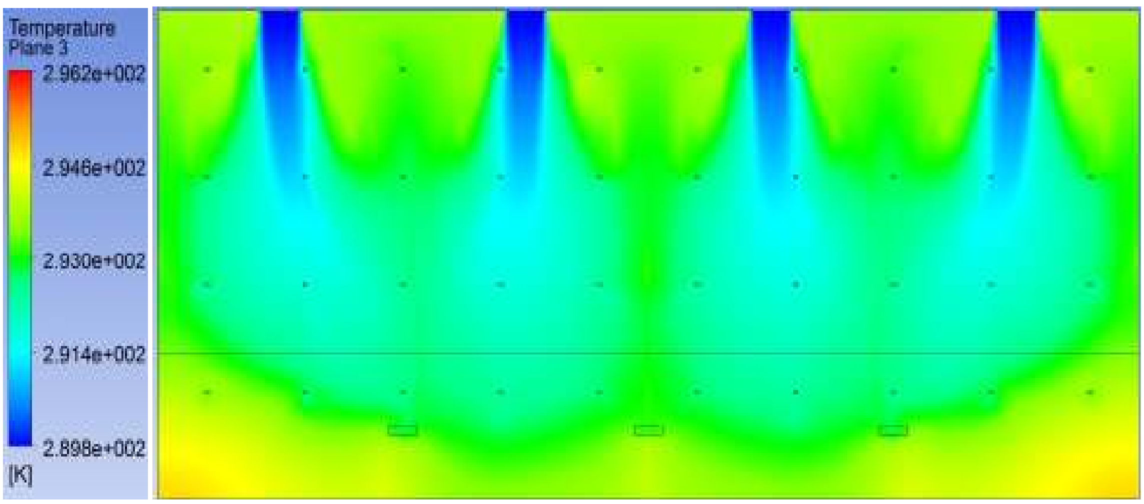



*Figure 9-1 Configuration#1. Air temperature field inside the underfloor plenum for the 25% of peak load.*

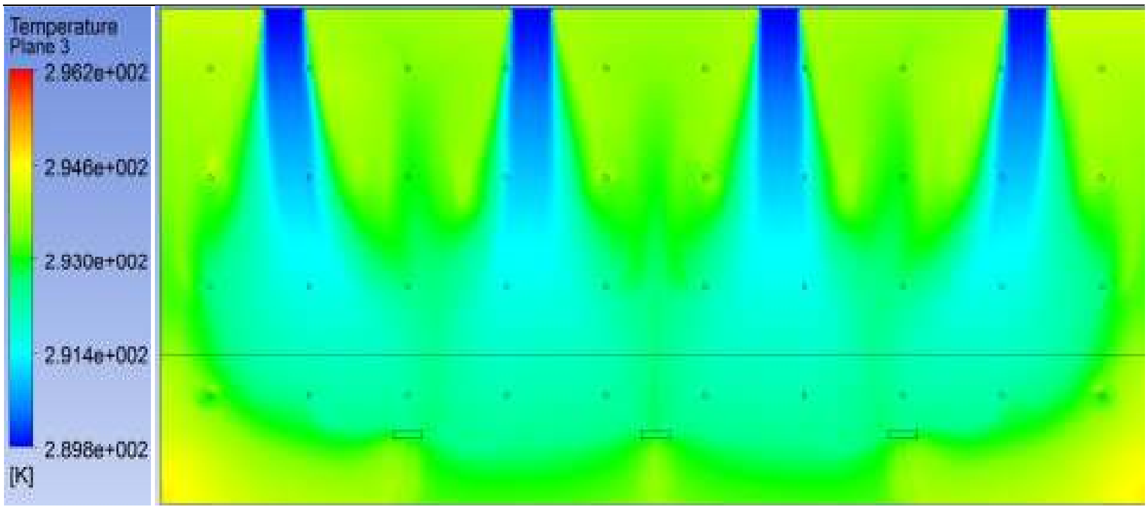



*Figure 9-2 Configuration#1. Air temperature field inside the underfloor plenum for the 50% of peak load.*

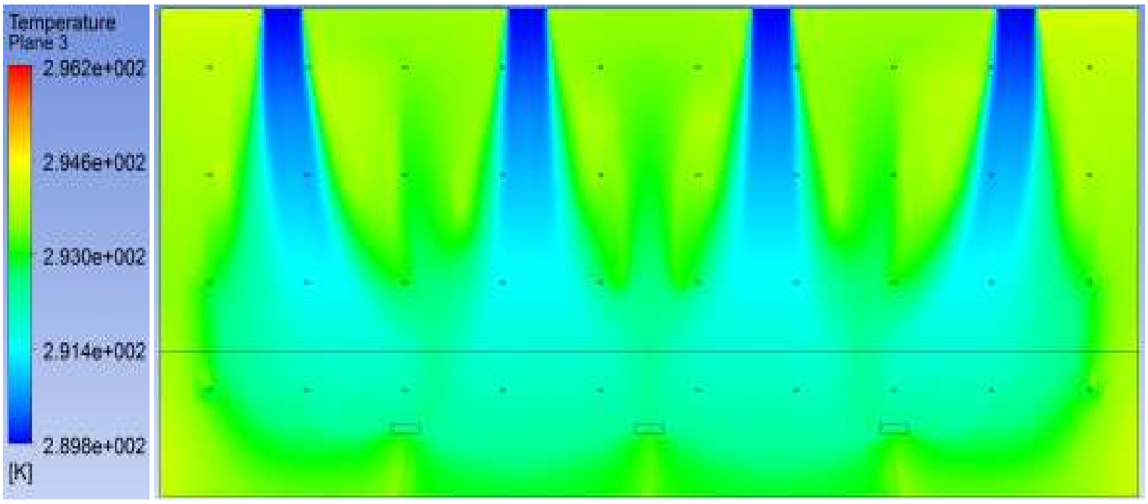



*Figure 9-3 Configuration#1. Air temperature field inside the underfloor plenum for the 75% of peak load.*

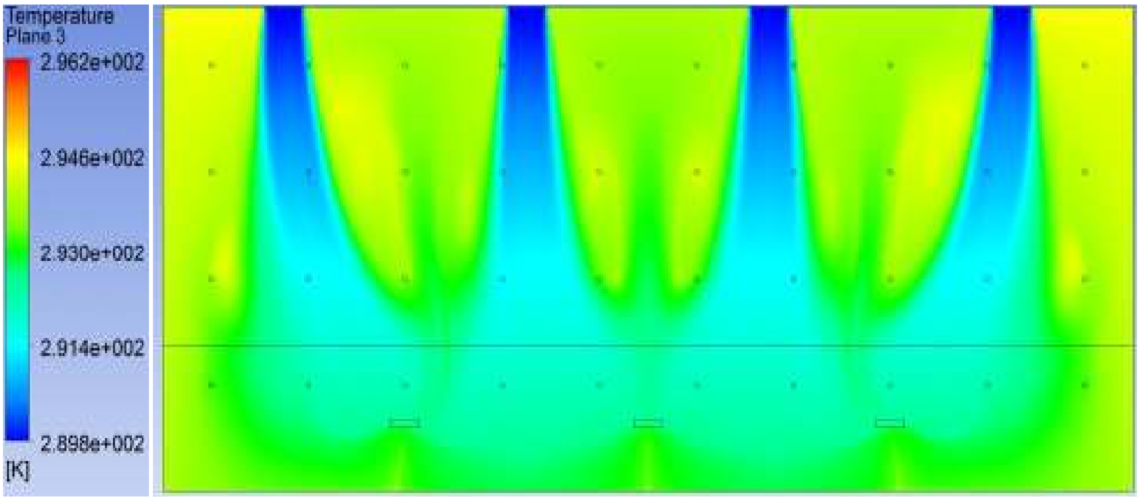



***Figure 9-4 Configuration#1. Air temperature field inside the underfloor plenum for the 100% of peak load.***

The big airflow values through the three fan coils strongly influence the airflow pathway into the plenum. This phenomena is clearly visible also in the following four figures.

The figures from Figure 9-5 to Figure 9-8 represent the air velocity field in the underfloor plenum for the configuration#1

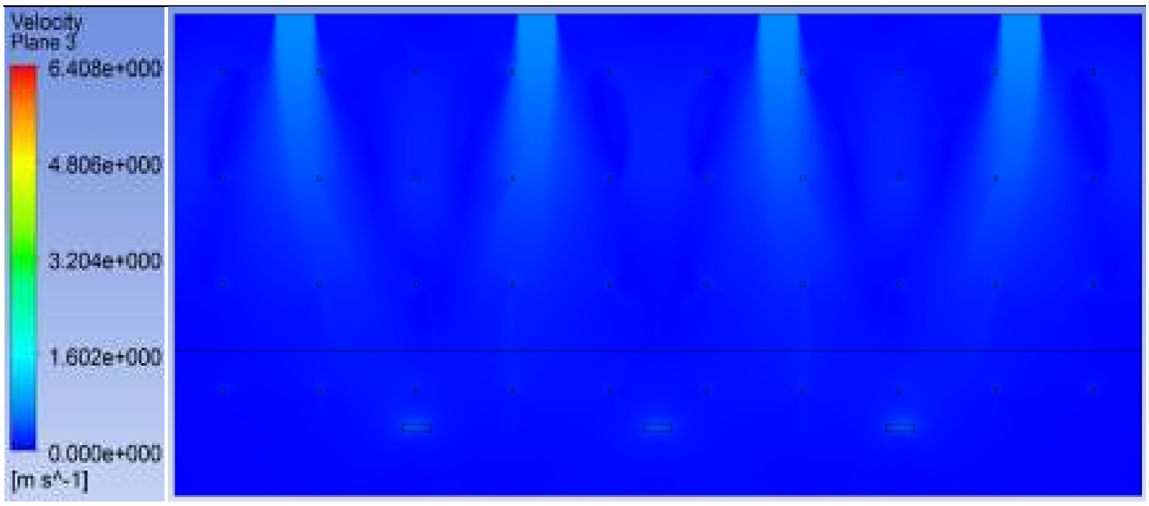



***Figure 9-5 Configuration#1. Air velocity field inside the underfloor plenum for the 25% of peak load.***

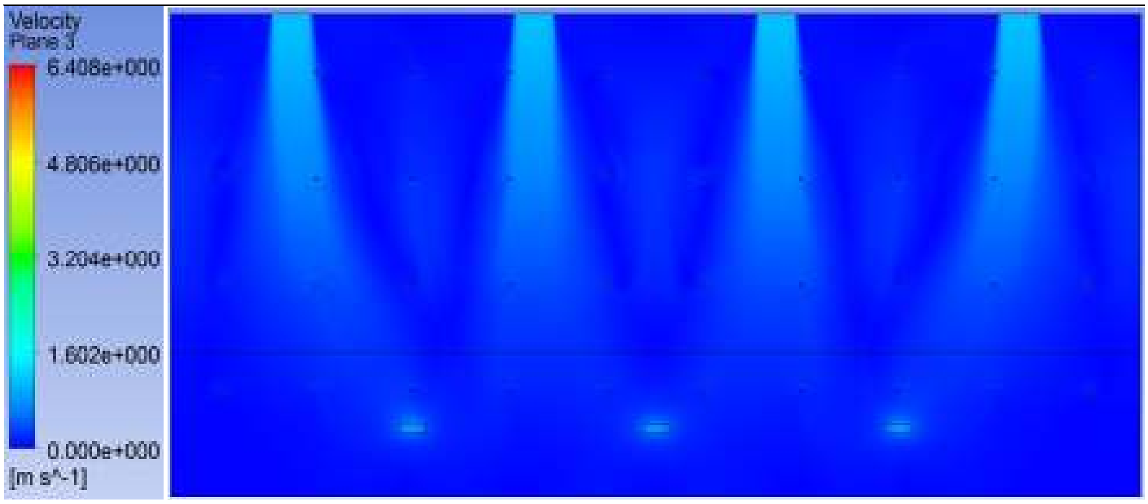



***Figure 9-6 Configuration#1. Air velocity field inside the underfloor plenum for the 50% of peak load.***

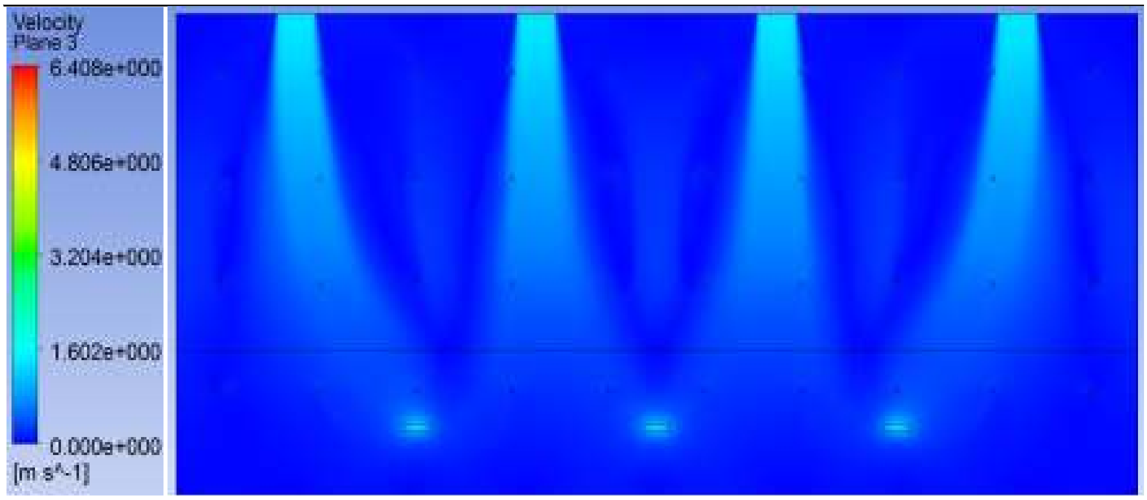



***Figure 9-7 Configuration#1. Air velocity field inside the underfloor plenum for the 75% of peak load.***

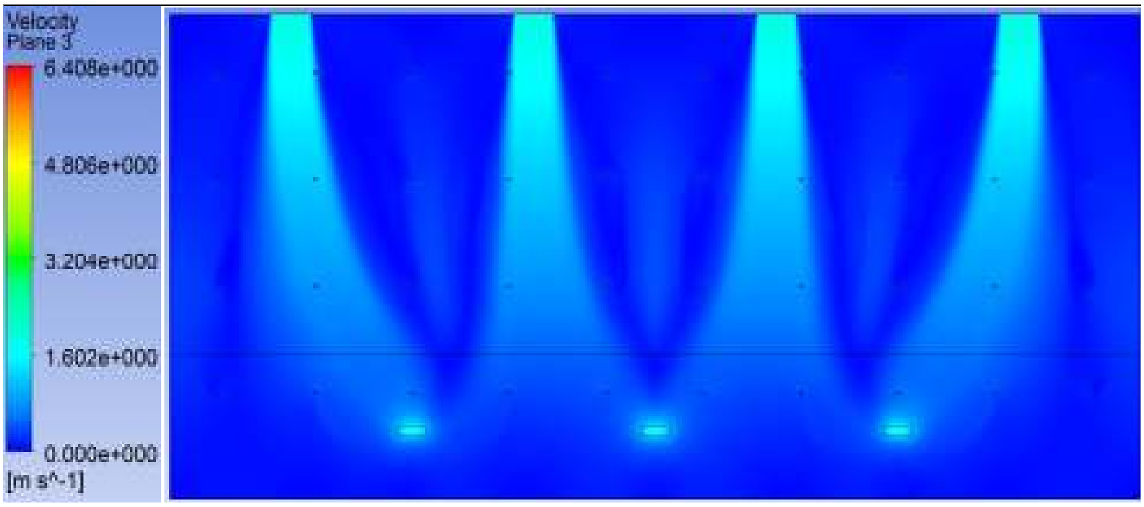



***Figure 9-8 Configuration#1. Air velocity field inside the underfloor plenum for the 100% of peak load.***

## 9.2 Configuration#2, temperature and velocity fields

In the images from Figure 9-9 to Figure 9-12, the underfloor temperature field for the configuration#2 for the four different load conditions, are presented.

The airflow pathway in the underfloor plenum is more complicated for the configuration#2 than for the configuration#1. In the following pictures it is possible to appreciate the unique feature of the vented fabric duct to distribute fresh air through the small orifices along the duct sides. Using ductwork, fabric or sheet metal, lets the system to deliver fresh air directly in the location where it is needed most, even for the lowest load unlike the previous model.

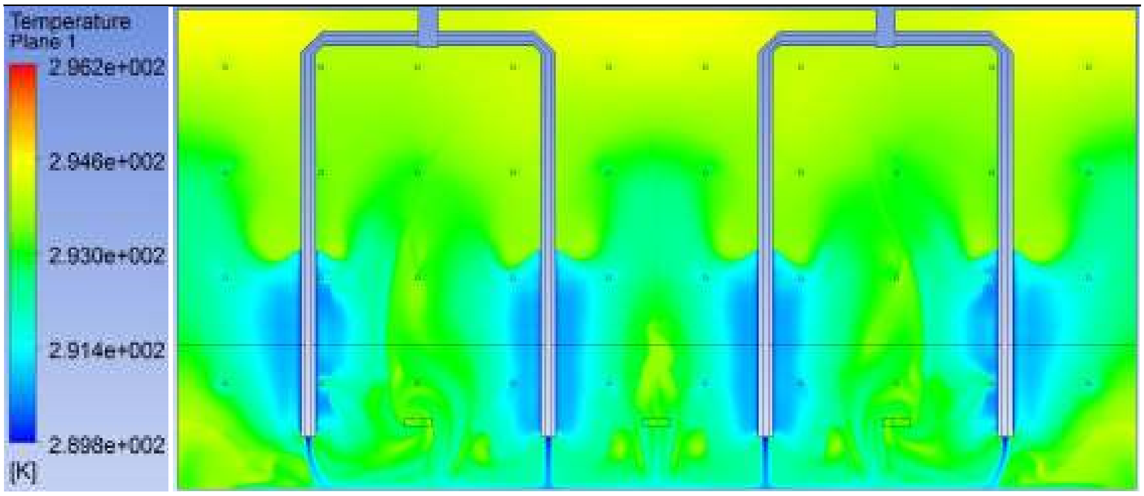



***Figure 9-9 Configuration#2. Air temperature field inside the underfloor plenum for the 25% of peak load.***

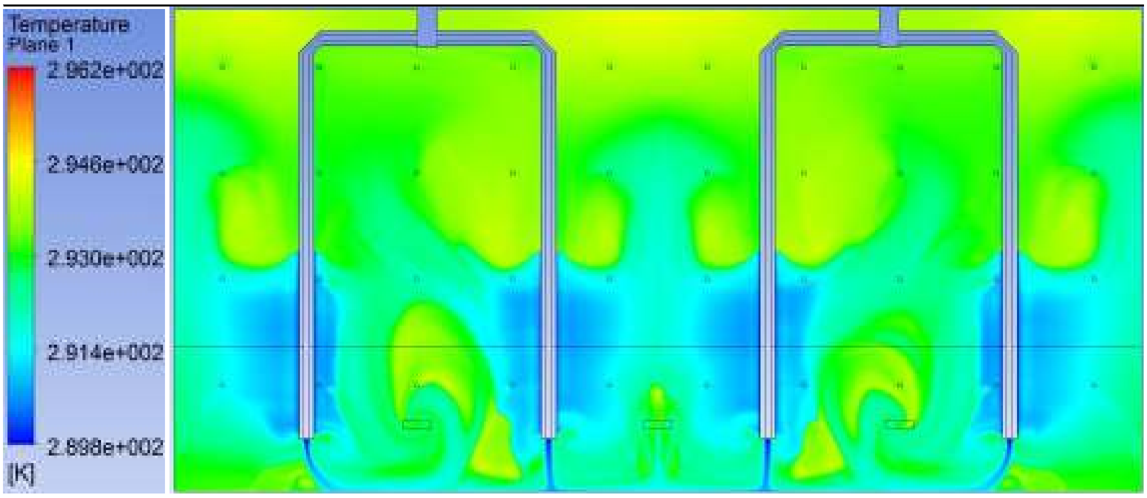



***Figure 9-10 Configuration#2. Air temperature field inside the underfloor plenum for the 50% of peak load.***

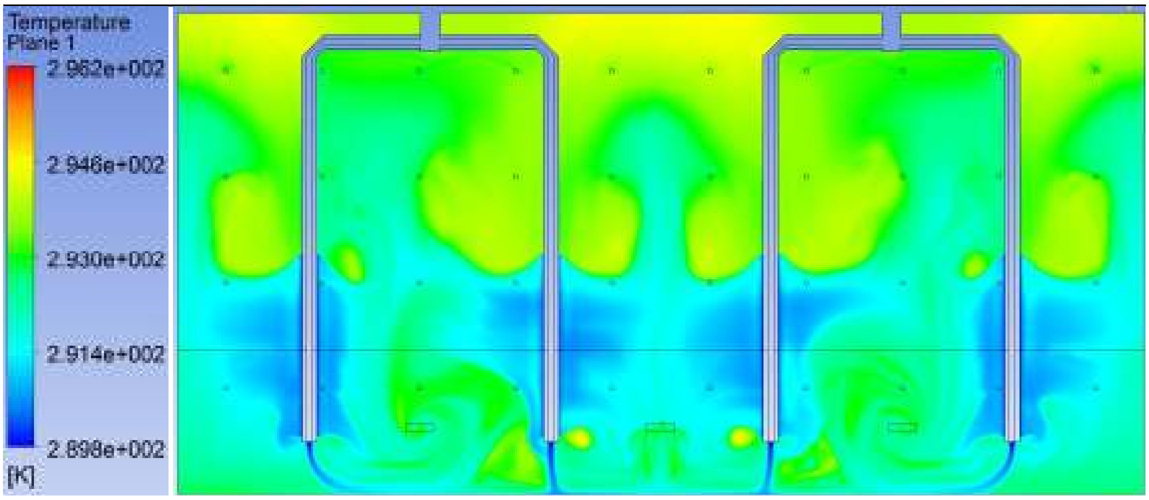



***Figure 9-11 Configuration#2. Air temperature field inside the underfloor plenum for the 75% of peak load.***

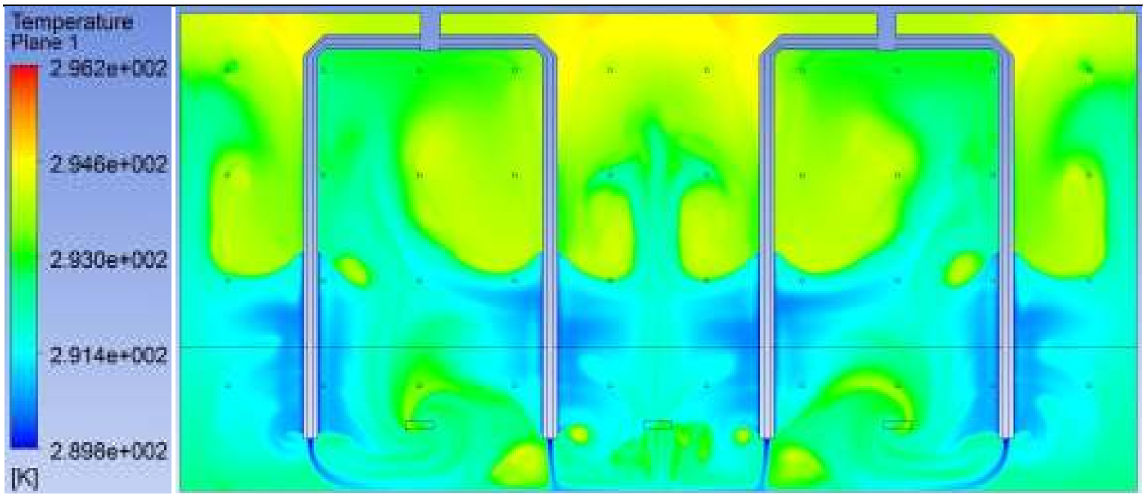



***Figure 9-12 Configuration#2. Air temperature field inside the underfloor plenum for the 100% of peak load.***

From Figure 9-13 to Figure 9-16 the air velocity field in the underfloor plenum for the configuration#2 are shown.

It is interesting to notice how the airflow from the end caps goes directly to feed the fan coils.

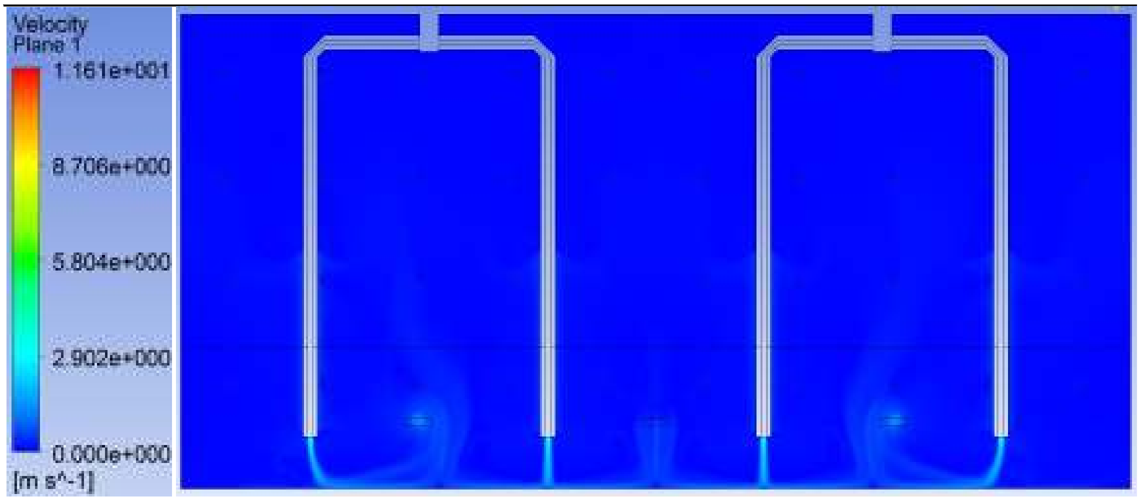



***Figure 9-13 Configuration#2. Air velocity field inside the underfloor plenum for the 25% of peak load.***

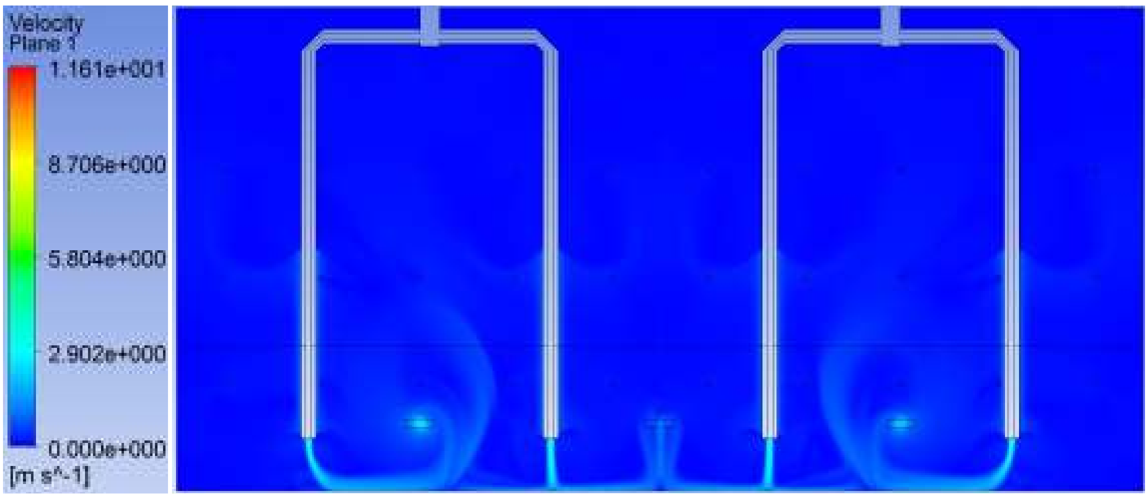



***Figure 9-14 Configuration#2. Air velocity field inside the underfloor plenum for the 50% of peak load.***

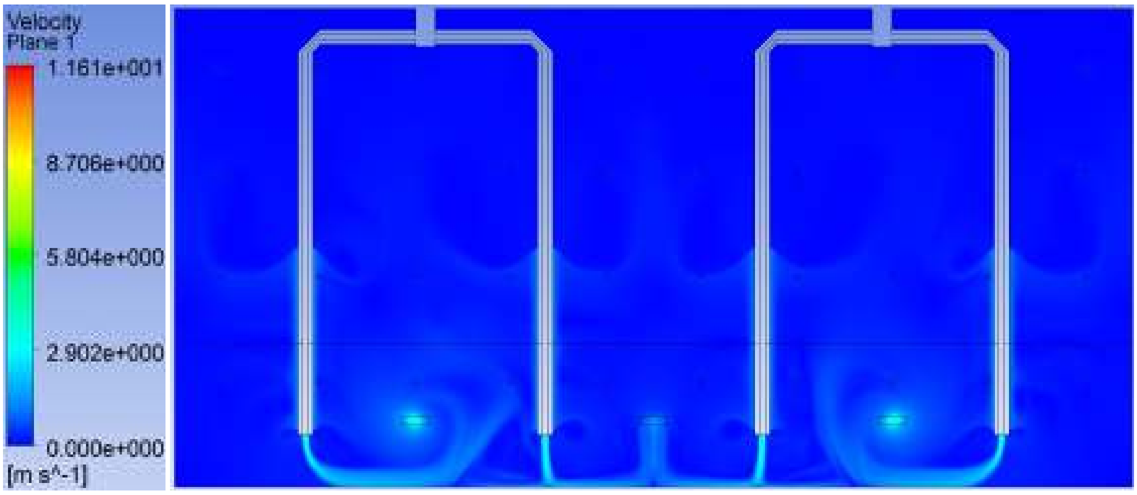



***Figure 9-15 Configuration#2. Air velocity field inside the underfloor plenum for the 75% of peak load.***

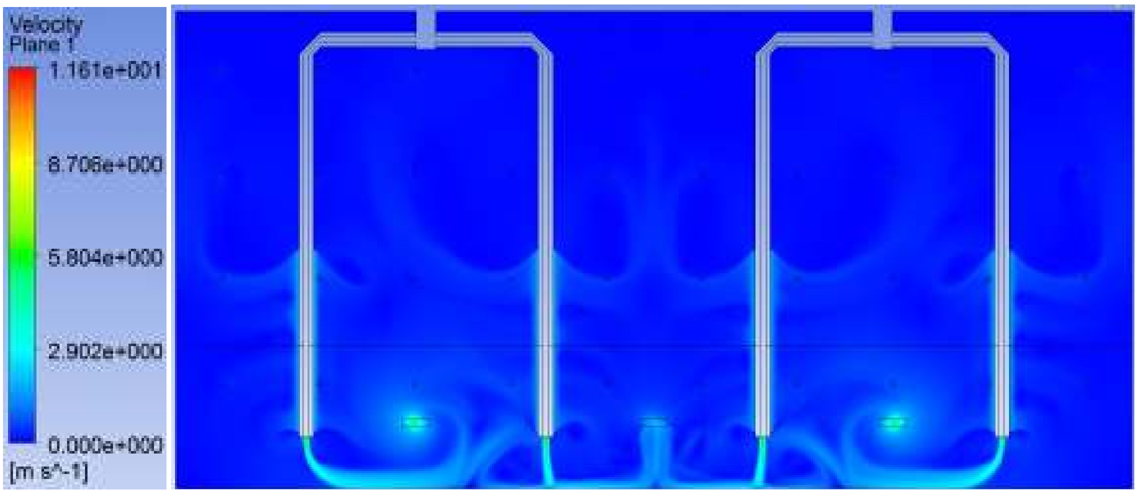



***Figure 9-16 Configuration#2. Air velocity field inside the underfloor plenum for the 100% of peak load.***

To sum up, in the configuration#1 it is desirable to have a strong air velocity to deliver air in the perimeter, but this for the configuration#2 it is not necessary because the fresh air is directly delivered in the perimeter. For the configuration#2 it is desirable to have air velocity as low as possible, in order to reduce the convective heat transfer along the slab and along the ceiling. In fact the lower is the air velocity in the underfloor plenum, and the lower will be the surface heat transfer coefficients, and the lower is the thermal decay. Hence, using the flexible duct to distribute air in the underfloor plenum could reduce the thermal decay besides reducing the air temperature for the perimeter diffusers.

## 9.3 CFD results analysis: perimeter temperatures

At this point with the collected CFD data it is possible compare the two models. The comparison is done in terms of average perimeter and interior temperatures for the air through the diffusers. The presented values are mass-flow average values. To estimate the mass-flow average temperature through a diffuser the following equation was used:

$$\bar{T}_i = \frac{\left(\sum_{j=1}^{m} \dot{m}_j \cdot T_j\right)}{\dot{m}_i}$$

***Equation 11***

where $\bar{T}_i$ is the mass-flow average temperature estimated for the diffuser *i*; $\dot{m}_j$ is the air mass-flow through the single cell *j* that constitute the mesh for the diffuser area, and $T_j$ is the mass-flow temperature for the same cell; $\dot{m}_i$ is the total mass-flow through the diffuser *i*.

To estimate the mass-flow perimeter and interior temperature the following equation was used:

$$\bar{T}_{int(perim)} = \frac{\left(\sum_{i(p)=1}^{30(10)} \dot{m}_{i(p)} \cdot \bar{T}_{i(p)}\right)}{\dot{m}_{int(perim)}}$$

***Equation 12***

where *i,* which goes from 1 to 30, is the interior diffuser number, than *p* as the same meaning but for the perimeter, and it goes from 1 to 10.

The previous equation estimates the mass-flow average temperature only for the diffusers, but in the perimeter there are also the three fan coils. For this reason to the Equation 12 the terms relative to the fan coils were added. The average mass-flow temperature for every fan coil was estimated applying the Equation 11 for the fan coil outlet surfaces.

In Figure 9-17 it is shown the mass-flow average perimeter temperature for the two configurations. The same figure shows that, as anticipated above, for the configuration#1 the perimeter temperature rises with the load decreasing, than for configuration#2 in which case the temperature is almost stable. It is also interesting to notice that the perimeter air temperature for the configuration#2 is always lower than that for the configuration#1, except for 100% load case, in which case the difference is negligible. The biggest difference is represented by the 25% load case. Under this condition the difference between the two configurations is 0.4 K.

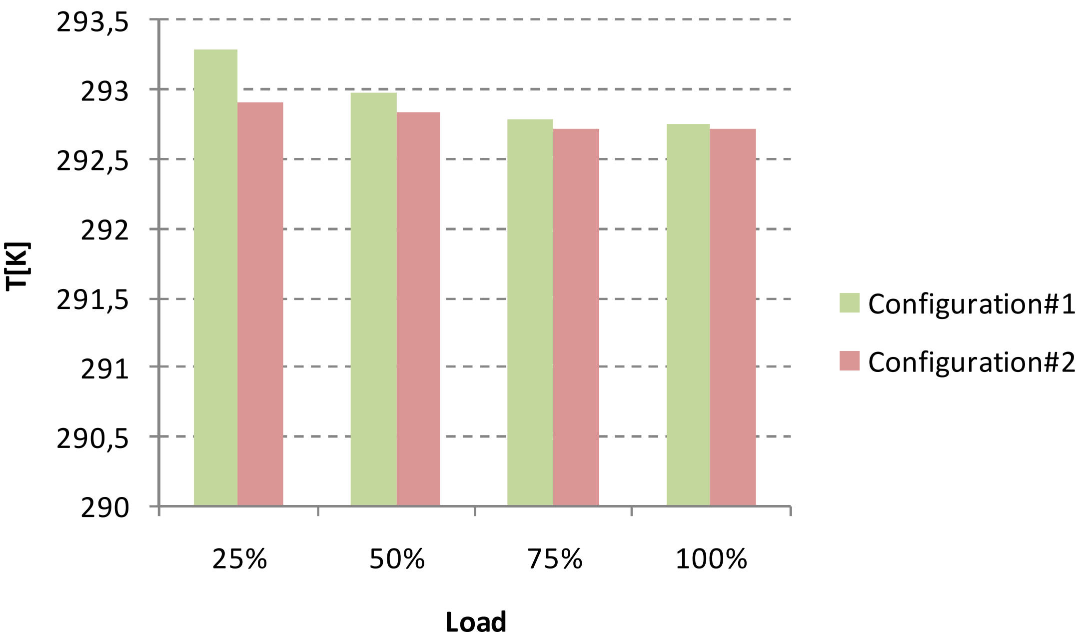



*Figure 9-17 Perimeter mass-flow average temperature comparison.*

In **Errore. L'origine riferimento non è stata trovata.** and Figure 9-19 the perimeter temperatures and interior temperatures for the two configurations are presented. Comparing the two temperatures for the same configuration under different load conditions, it is possible to notice, in Figure 9-18, an interesting temperature inversion from 25% load conditions to 100% load conditions, for the configuration#1. This phenomena it is not present for the configuration#2 (Figure 9-19), for this configuration, where for every load conditions, the perimeter temperatures are always lower than the interior temperatures. In the practice this last condition is the most desirable to improve the thermal comfort in the conditioned space, and to reduce the HVAC energy demand.

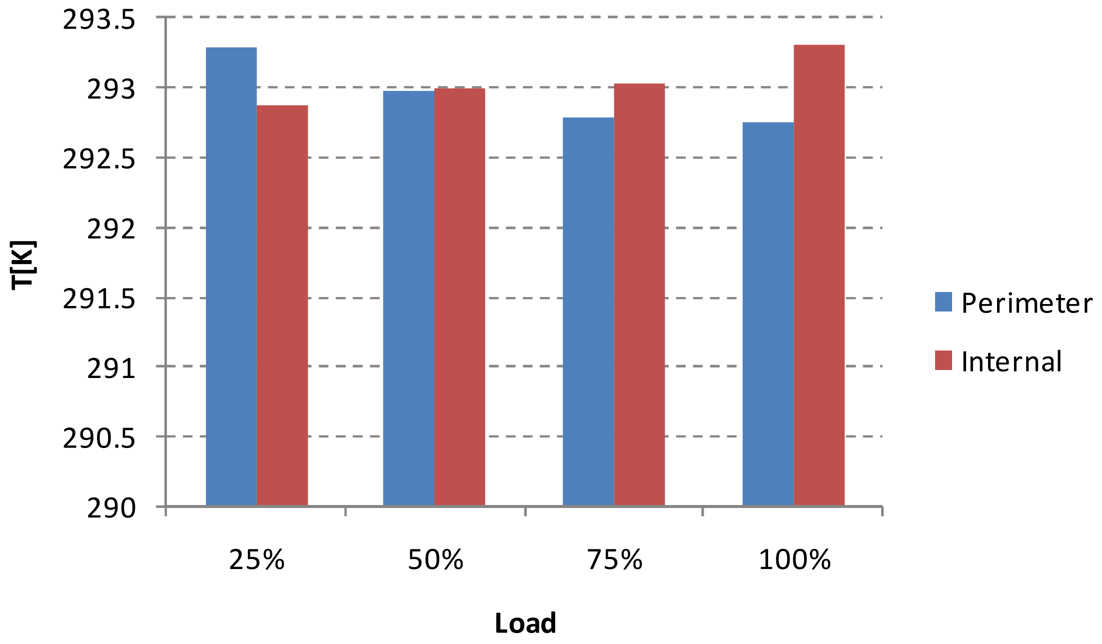



*Figure 9-18 Configuration#1: perimeter temperatures and interior temperatures.*

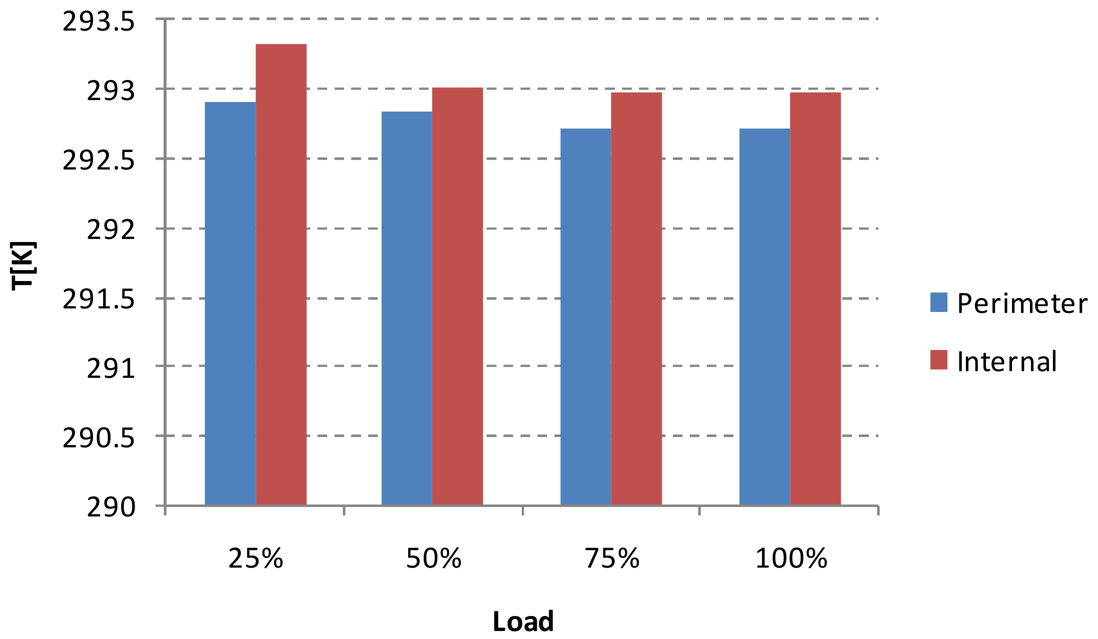



*Figure 9-19 Configuration#2: perimeter temperatures and interior temperatures.*

## 9.4 CFD results analysis: thermal decay

Using the CFD results it is possible to analyze the thermal decay phenomena, and see if there is any difference between the two configurations.

The thermal decay inside the underfloor plenum it is due to heat exchange between air and plenum surfaces (slab, and bottom surface of raised floor). Few pictures, which can help to visualize the ceiling heating exchange, are reported below. The two configurations are shown alternately, to help visualize the difference between the models better.


***Figure 9-20 Configuration#1, 25% load conditions: ceiling heat flux.***

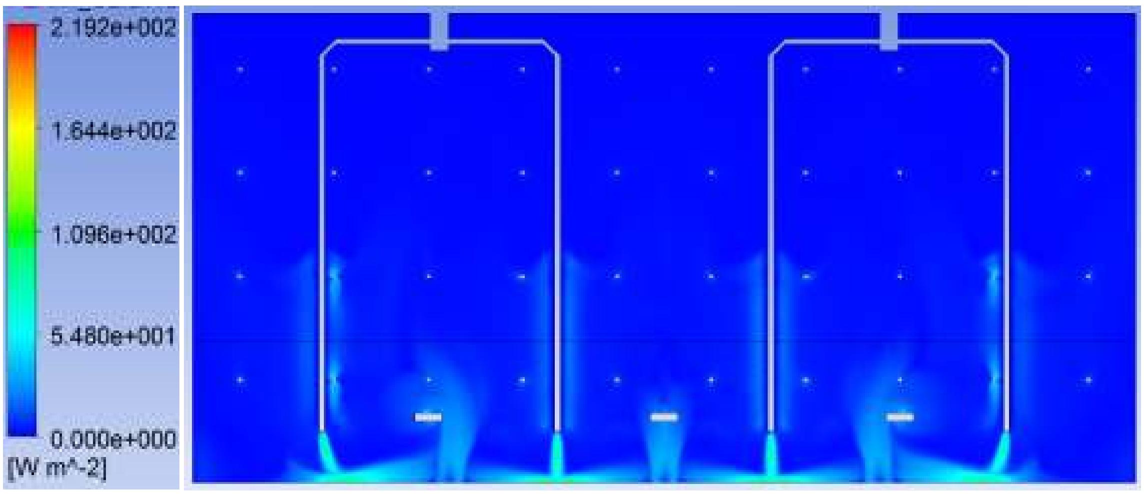



***Figure 9-21 Configuration#2, 25% load conditions: ceiling heat flux.***


*Figure 9-22 Configuration#1, 50% load conditions: ceiling heat flux.*

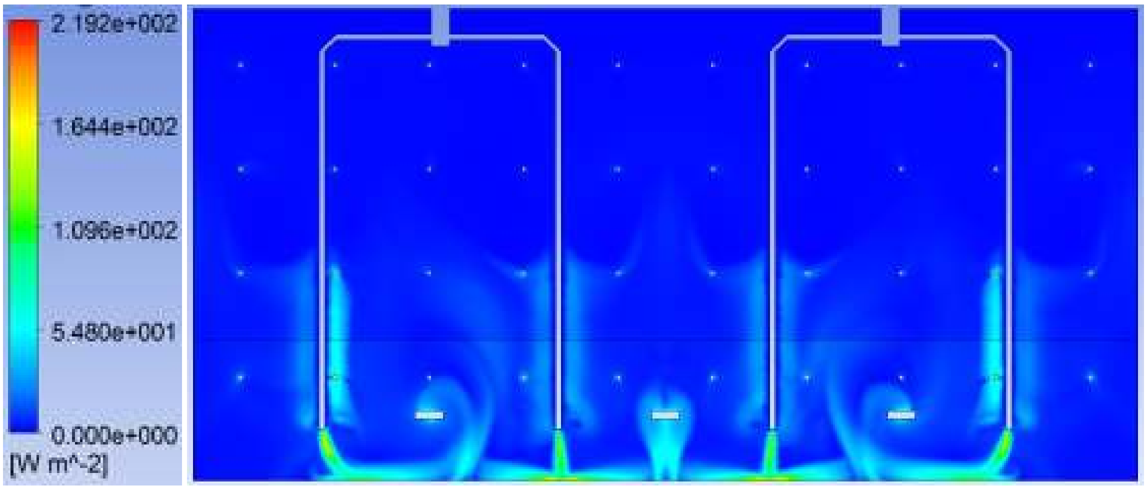



*Figure 9-23 Configuration#2, 50% load conditions: ceiling heat flux.*

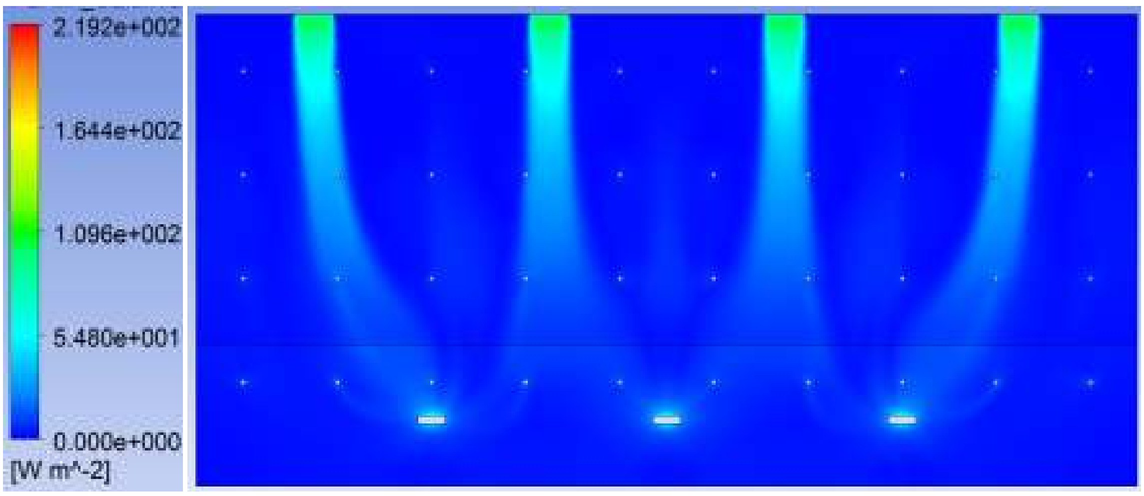



*Figure 9-24 Configuration#1, 75% load conditions: ceiling heat flux.*

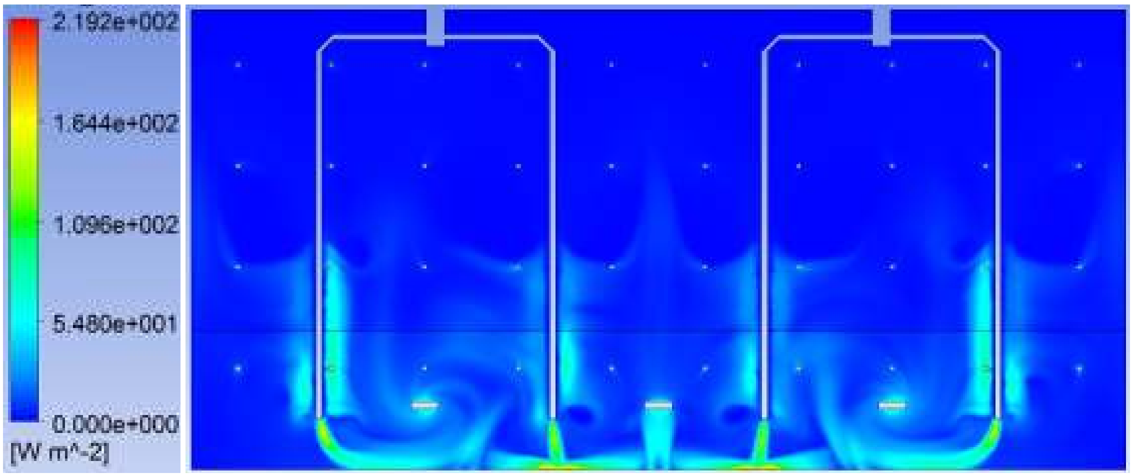



***Figure 9-25 Configuration#2, 75% load conditions: ceiling heat flux.***

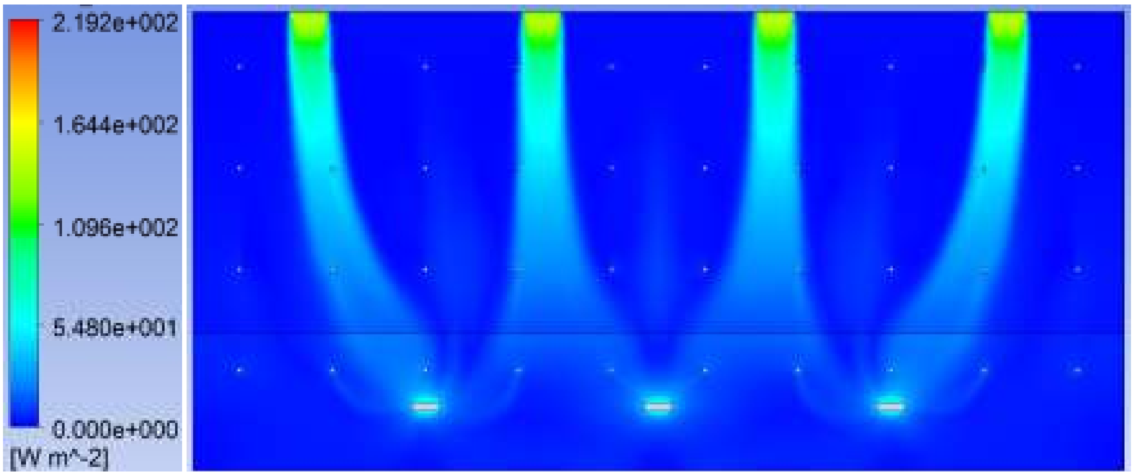



***Figure 9-26 Configuration#1, 100% load conditions: ceiling heat flux.***

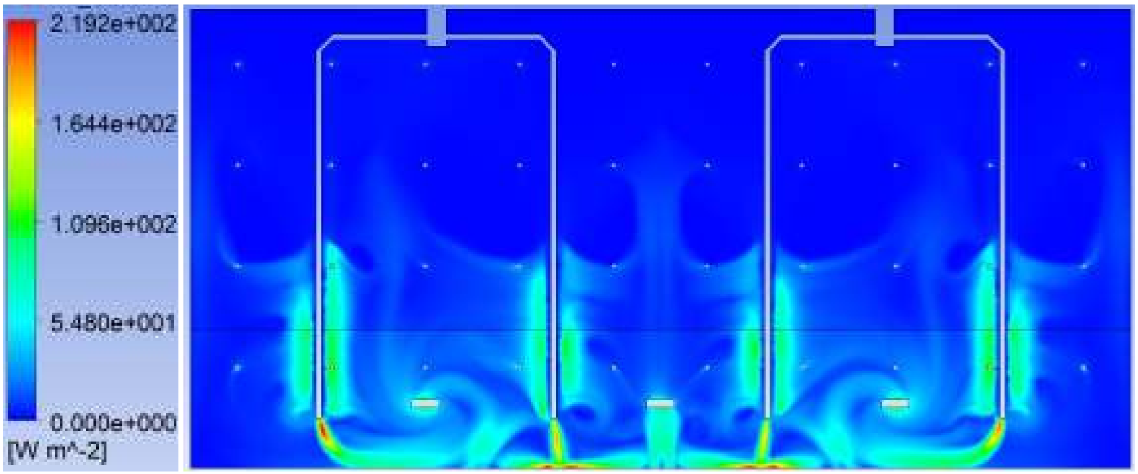



***Figure 9-27 Configuration#2, 100% load conditions: ceiling heat flux.***

Analyzing these last figures (from Figure 9-20 to Figure 9-27), it is possible to recognize that the maximum heat exchange takes place in two different parts of the plenum for the two configurations. For the configuration#1 it is located in the interior part of the plenum, than for the configuration#2 it occurs in the perimeter. For a typical underfloor configuration, like this, this is an obvious phenomena but it must be highlighted. It is interesting because the higher airflow demand is located in the perimeter so, for the configuration#1 a big amount of fresh air goes through the interior plenum to the perimeter, than for the configuration#2 where the same big amount of fresh air is delivered directly into the perimeter near to the outlets, but on the other hand the perimeter surfaces have higher temperatures than interior surfaces. So having the biggest amount of heating exchange concentrated in the perimeter could negatively influence the thermal decay. At this point it is useful to analyze the total air heating exchange to see if these phenomena have some influence on the total heating exchange.

In **Errore. L'origine riferimento non è stata trovata.** the comparison between the two model for the total heat exchanged is illustrated.

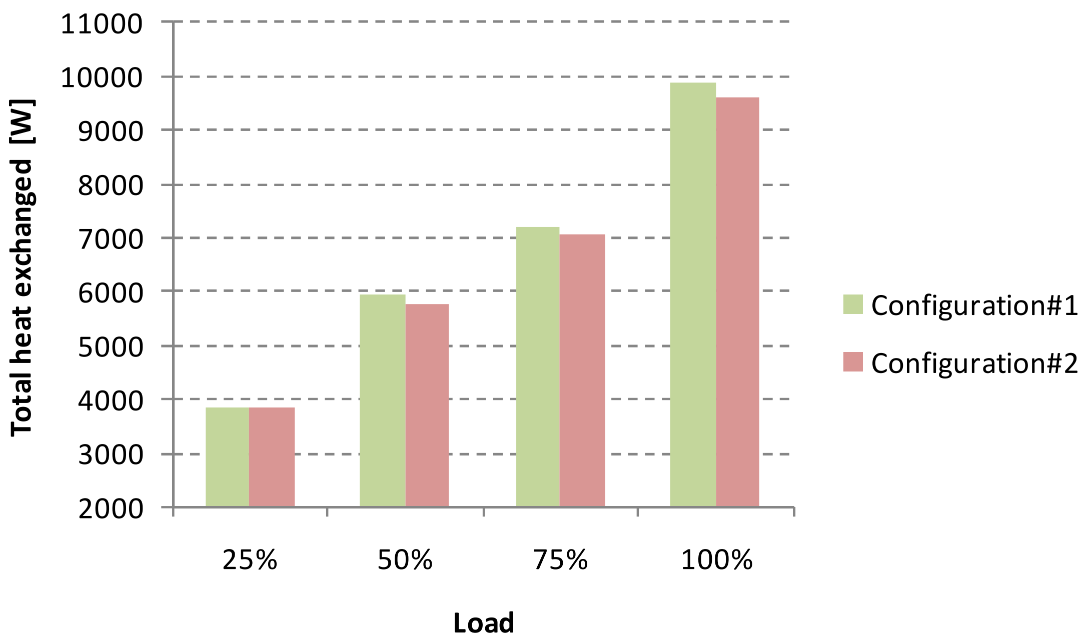



***Figure 9-28 Total heat exchanged comparison***

The heat power exchanged for the two configurations is almost the same for all the four load conditions. This could be read in two alternative ways: there is no correlation between inlet and outlet relative positions and the heating exchange, or the two phenomena balance each other. Some other simulations should be done to better investigate these phenomena and their influence on the thermal decay. The biggest heat flux difference was observed for 100% load case, and it is equal to 2.9% of the total heat flux.

To better understand the phenomenon a new model was implemented. In this model all air enters into the plenum through the vented ducts. Everything is still the same except that the end caps are solid. The new strategy was tested under max cooling load condition. This test was done to see if there is difference in terms of heat flux, if the relative position between plenum inlet and outlet is changed. The results are presented below:

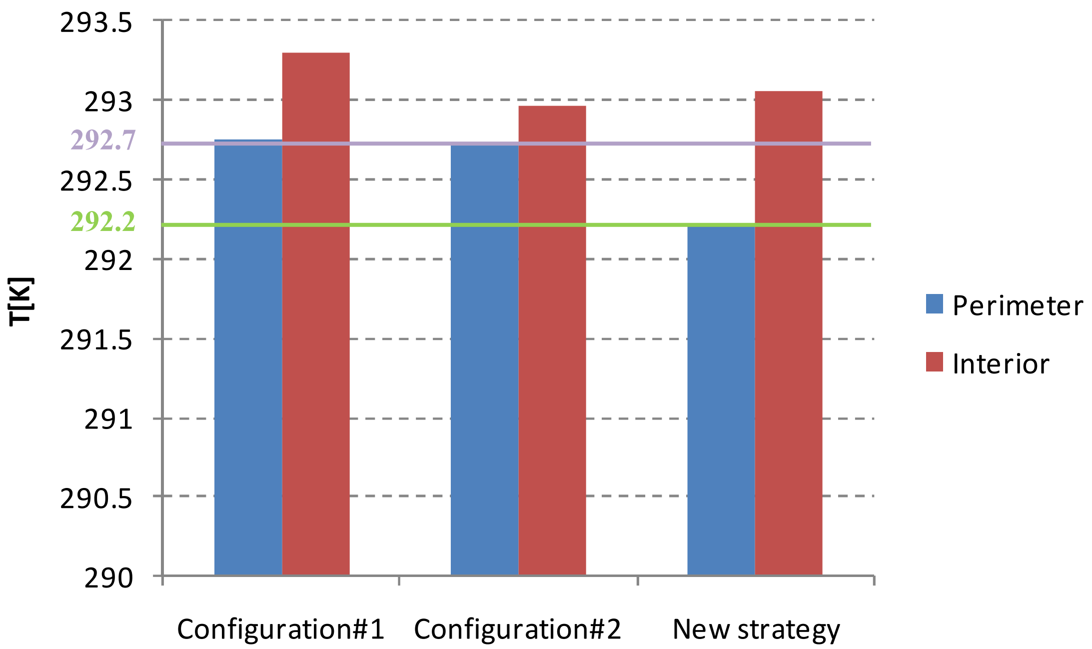



***Figure 9-29 Perimeter and interior air temperature comparison***

As it can be seen in **Errore. L'origine riferimento non è stata trovata.** the new strategy configuration presents the lowest perimeter temperature, but also the interior temperature is quite low, almost the same than configuration#2. This is a very interesting result because it goes in the right direction to achieved a lower perimeter temperature without compromising the interior temperature. Even more interesting it is the heat flux analysis presented in Figure 9-30. The total heat flux in this new configuration is 15%

less than configuration#1. This is interesting and it can be explained by different phenomena:

- delivering air only through the vented ducts reduces the air velocity in the plenum perimeter and so it reduces the wall heating exchange. The average air velocity measured in a plane placed at 2 cm from the underside surface of raised floor ( same for the slab surface) has the following values respectively for configuration#2, and new configuration: 5.06 m/s and 4 m/s. So the lower perimeter air velocity and relative low perimeter wall heat flux can explain the difference between configuration#2 and new configuration;
- for the new configuration the distance between plenum inlet and the main outlets ( the fan coil units) is lower compared with all the other configuration. This can explain the thermal decay reduction comparing with configuration#1

Analyzing this new configuration it is possible explain why the previous analysis does not present any significant difference in terms of heat flux between configuration#1 and configuration#2. The lower distance between plenum inlets and main outlets in configuration#2 is counterbalanced by an higher air velocity close to the perimeter wall surfaces, which results in an higher wall heat flux, also because the perimeter surface temperatures are higher. Figure 9-31 and Figure 9-32 show the temperature distribution in the underfloor plenum, and wall heat flux for the floor surface (same for the slab surface).

As experimental tests have shown, using solid caps to close the ducts can cause a not negligible pressure rising in the HVAC system. A good solution, to keep under control the pressure in the HVAC system, and to reduce the thermal decay, could be using vented ducts with an higher orifices diameter. Few more experimental tests and CFD analysis should be done to better investigate this phenomenon.

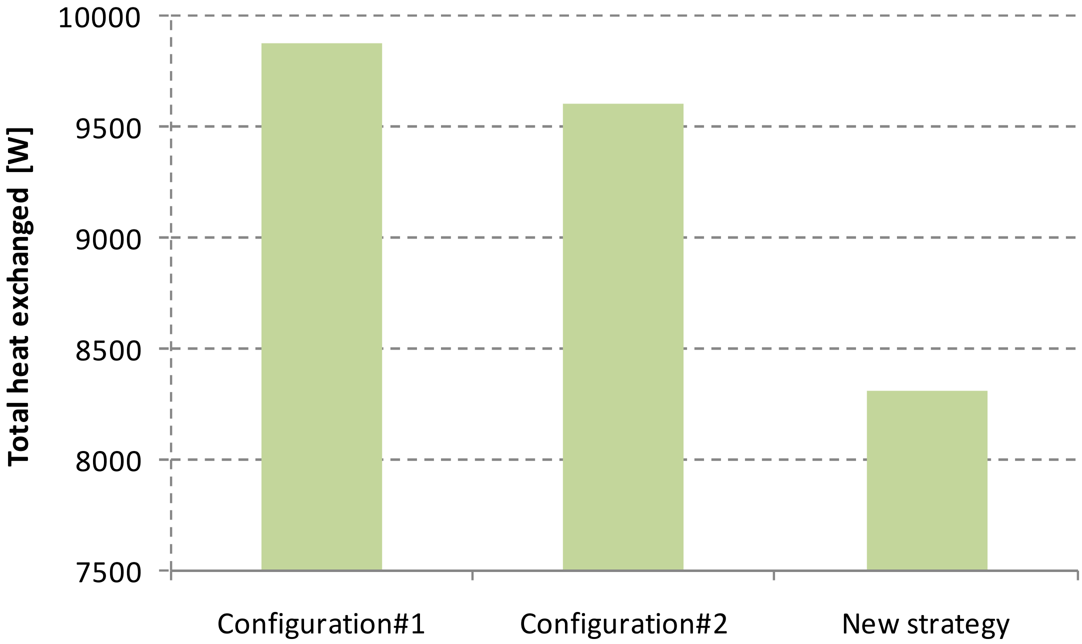



***Figure 9-30 Total heat exchanged comparison***

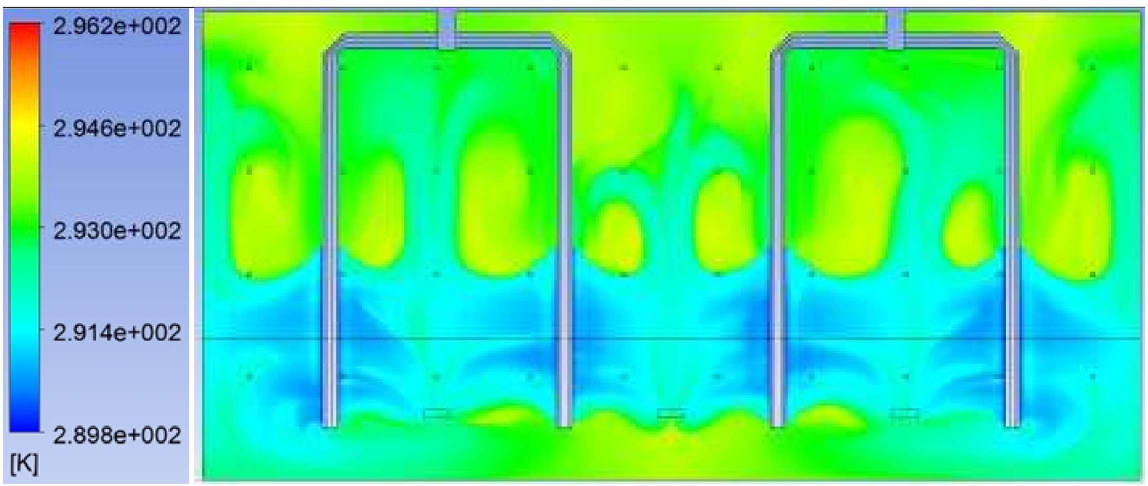



***Figure 9-31 New model: temperature field***

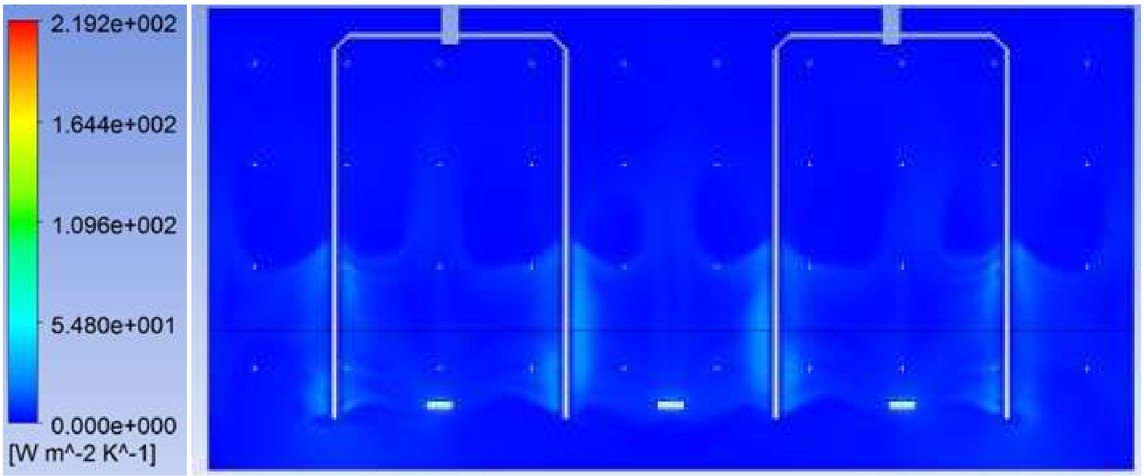



***Figure 9-32 New model: wall heat flux***

Few words should be spent on the position of the three fan coils. As it can be seen in the previous figures (i.e. Figure 9-4), when the air flow delivered through a fan coil is high, this strongly influences the plenum airflow pathway. A wiser position of the fan coil could improve the airflow pathway in the plenum. For example placing four fan coil in line with the plenum inlet, for the configuration#2, could improve the fan coils feeding. It should be interesting to analyze in future how the fan coil positions and numbers can influence the airflow pathway and heating exchange in the plenum.

# 10 Conclusions

The aim of this work is to analyze the use of ductwork (flexible/fabric or rigid) to improve the air distribution in underfloor plenum systems, with particular attention to analyze the capability of fabric ductwork systems to reduce air temperatures in the plenum perimeter. Already during the tests in the underfloor plenum facilities, it was possible to notice some characteristics of these kinds of systems. The tests result bear evidence of a lower average air temperature for the perimeter diffusers, suggesting the system capability to reduce air temperature in the perimeter, but it has also evidenced an important rising of pressure in the HVAC system for the case with the vented duct and solid end-cap. This is not insignificant at all as the fan, in the HVAC system, consumes much more energy to deliver fresh air into the plenum. This phenomenon was not encountered for the configurations without the end cap. So it is possible deduce a first advise for the practice of employing ducts: it must always take into consideration a partial opening of the end cap or an appropriate orifices dimension and number to reduce the pressure in the HVAC system and then fan energy demand.

The tests results have shown the importance of relative position between air plenum inlets and diffusers. In fact if a diffuser is too close to a strong plenum airflow pathway, the airflow through the diffuser will be sensibly reduced. This phenomenon is visible in Figure 4-17 for the diffusers number five and six. The airflows of these two diffusers are strongly influenced by the strong air flux into the plenum due by the proximity of the plenum air inlet. It registered an airflow reduction for these two diffusers by around 20-25% compared to the other two perimeter diffusers. Also related to this, it was observed during the CFD modeling work, the importance of modeling the diffuser part located in the underfloor plenum. This part and his orientation can influence the airflow through the diffuser.

The CFD modeling work, combined with the experimental work, has produced a CFD model that is able to predict the behavior of a real plenum with a very good level of confidence, and that can be used to investigate the behavior of these systems under different operative conditions.

The main conclusions are related with the second part of the CFD simulation work. In that part the CFD fabric model was developed using the test results, to compare this technical solution with another well diffused solution used to reduce air temperature through the perimeter diffusers. It was proved that delivering air into the plenum with a high momentum to reach the perimeter, loses its capability to reduce perimeter air temperature under partial load conditions. More than this, under part load conditions (25%) there is a reversed result, the perimeter air is warmer than interior air. That could cause some problem in terms of controlling room air temperature and comfort level. On the contrary, using flexible (or rigid) ducts to deliver fresh air into the plenum perimeter, keeps the proportion between interior air and perimeter air always the same through every partial load conditions (see Figure 9-19), with perimeter air temperature values always lower than interior air temperature. Also comparing the two different solutions under the same load condition, has shown that using ductwork produces always lower perimeter airflow temperatures (Figure 9-17). It was also proved that it is important to reduce the distance between the plenum inlets and the main outlets, using ductwork or other technical solution. The importance of a low air velocity through the plenum inlet, to reduce the wall heat flux, was also proved. Delivering fresh air into the plenum perimeter with high momentum could reduce the perimeter temperatures but not the heat flux, while using a well designed ductwork reduces the heat flux and, as a consequence, the perimeter temperatures. This is a very important conclusion that, combined with the behavior predictability, should lead to use ductwork in underfloor air distribution system, in particular for retrofit applications.

# 11 References

1. **Bauman, Fred S.** *Under Floor Air Distribution (UFAD) Design Guide.* s.l. : ASHRAE, December 2003.

2. *The Underfloor air supply system-the European experience.* **Sodec, F. and Craig, R.** No. Part 2., s.l. : ASHRAE Transactions, 1990, Vols. 96, No. Part 2.

3. **Sodec, F.** *Air distribution systems report no. 3554A.* s.l. : Krantz GmbH & Co, September 1984.

4. *Task / ambient conditioning system in Japan.* **Tanabe, S.** San Francisco : s.n., May 1995. presented at Workshop on task / ambient conditionig system in commercial buildings.

5. *Alternative Air Conditioning Technologies:Underfloor Air Distribution (UFAD).* **Webster, T.** 6, s.l. : Energy Engineering, 2005, Vol. 102.

6. *Threshols for indoor thermal comfort and perceived air quality.* **Zhang, H., Arens, E. and Pasut, W.** Windsor, UK : s.n., April 2010. Adapting to Change: New Thinking on Comfort.

7. http://www.cbe.berkeley.edu/underfloorair/thermalcomfort.html. [Online]

8. *Underfloor Air Distribution: Thermal Stratification.* **Webster, T., Bauman, F. e Reese, J.** 5, pp. 28-36, s.l. : ASHRAE Journal, May 2002, Vol. 44.

9. *Modelling an underfloor air distributin system.* **Lin, Y.-J. and Linden, P.F.** Copenhagen : s.n., September 2002. ROOMVENT 2002.

10. **ASHRAE.** *ANSI/ASHRAE Standard 55-2010"Thermal environmental conditions for human occupancy".* Atlanta : American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc., 1992.

11. *Underfloor Air Distribution (UFAD).* **Bauman, Fred.** Venice : s.n., 2008. Workshop "Integrated design of HVAC systems in buildings".

12. *Impact of the thermal decay on the underfloor air distribution (UFAD) system performance.* **Lee, Kwang Ho, et al.** s.l. : Under review, 2010.

13. *Heat Transfer Pathways in Underfloor Air Distribution (UFAD) Systems.* **Bauman, Fred, Hui, Jin e Webster, Tom.** s.l. : American Society of Heating, Refrigerating and Air-Conditioning Engineers Inc., 2006. ASHRAE Transactions Vol. 112, Part 2.

14. *Testing and Modeling of Underfloor Air Suply Plenum.* **Hui, Jin, Bauman, Fred e Webster, Tom.** s.l. : ASHRAE Transactions, Vol. 112, Part 2, 2006.

15. **DUCTSOX CORP., Dubuque ,IA.** *http://www.ductsox.com/.* [Online]

16. http://www.cbe.berkeley.edu/research/pdf_files/ufaddesigntoolnotes.pdf. *http://www.cbe.berkeley.edu.* [Online] 2010.

17. *Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 1- Summary of Prevalent Turbulence Models.* **Zhai, Z., et al.** 6, s.l. : HVAC&R Research, 2007, Vol. 13, p. 853-870.

18. *Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environment by CFD: Part 2- Comparison with Experimental Data from Literature.* **Zhang, Z., et al.** 6, s.l. : HVAC&R Research, 2007, Vol. 13, p. 871-886.

19. *Comparison of airflow and contaminant distributions in room with traditional displacement ventilation and under-floor air distribution systems.* **Lee, Kisup, et al.** 115(2), s.l. : ASHRAE Transactions, 2009.

20. *The numerical computation of turbulent flows.* **Launder, B.E. e Spalding, D.B.** 3, s.l. : Computer Methods in Applied Mechanics and Energy, 1974, p. 269-289.

21. *ANSYS CFX, Release 12.0, User's guide.*

22. **ANSYS, Inc.** *ANSYS CFX-Solver Theory Guide.* Southpointe, 275 Technology Drive, Canonsburg, PA 15317 : s.n., April 2009.

23. *The Design and Application of Upwind Schemes on Unstructured Meshes.* **Barth, T.J. e Jesperson, D.C.** 89, s.l. : AIAA Paper, 1989.

24. EnergyPlus Input Output Reference. The Encyclopedic Reference to EnergyPlus Input and Output. *http://www.energyplus.gov.* [Online] 2010.

25. **Schiavon, Stefano, et al.** *Development of a simplified cooling load design tool for underfloor air distribution systems.* 2010. Final Report to CEC PIER Program.

26. **J.Anderson, E. Dick, G. Degrez, R. Grundmann, J. Degroote, J. Vierendeels.** *Computational Fluid Dynamics.* s.l. : Springer, 2010.

27. *Experimental measurements and numerical simulations of particle transport and distribution in ventilated rooms.* **Zhang, Z. e Chen, Q.** 40, s.l. : Atmospheric environment, 2006, p. 3396-3408.

28. **ANSYS, Inc.** *Ansys CFX-Solver Modeling Guide.* Southpointe, 275 Technology Drive,Canonsburg, PA 15317 : s.n., April 2009.