Application of coupling of the human and clothing thermal system and computational fluids dynamics in the evaluation of energy and comfort in occupied spaces

Eusébio Conceição¹, Mª Inês Conceição², Mª Manuela Lúcio¹, João Gomes³ and Hazim Awbi⁴
¹Faculdade de Ciências e Tecnologia, Universidade do Algarve, Campus de Gambelas, 8005-139 Faro, Portugal  
²Instituto Superior Técnico, Universidade de Lisboa, Av. Rovisco Pais, 1049-001 Lisboa, Portugal  
³CINTAL, Campus de Gambelas, 8005-139 Faro, Portugal  
⁴School of Built Environment, University of Reading, Reading, RG6 6AW, United Kingdom

Abstract
This numerical work develops a HVAC system, based in a ceiling mounted localized air distribution system. This work presents and applies a numerical model that considers the coupling of the Computer Fluid Dynamics and Human Thermal Response. The coupling system, itself, generates all equations and space geometry and occupation presence and transfers the inputs/output between the Computer Fluid Dynamics and Human Thermal Response numerical models. The study is made in a virtual chamber occupied by twenty-four virtual occupants and twenty-four seats and equipped with a new ceiling mounted localized air distribution system. The inlet airflow and the outlet is located above the head levels. In the present study the thermal comfort level, the air quality level, the effectiveness and the ADI are calculated, in winter conditions. In accordance with the obtained results the ADI index increase slightly when the inlet air velocity increase.

Key Innovations
- Coupling of Computer Fluid Dynamics and Human Thermal Response;
- Equations generation;
- Evaluation of human thermal comfort and the indoor air quality;
- Evaluation of Draught Risk level;
- Calculation of Air Distribution Index.

Practical Implications
The Computer-Aided Design, the grid refined and the number of occupants are used to generate all integral and differential equations system. When the equations number increase the computational time increase.

Glossary
- \( N \) – Number of occupants;
- \( T_m \) (°C) – Body mean temperature;
- \( E_{HC} \) (%) – Effectiveness for heat removal;
- \( PPD \) (%) – Predicted Percentage of dissatisfied People;
- \( N_{TC} \) – Thermal Comfort Number;
- \( C \) (mg/m³) – Carbon dioxide concentration in the respiration area;
- \( E_{IAQ} \) – Effectiveness for contaminant removal;
- \( PD_{IAQ} \) – Predicted Dissatisfied people related with the Indoor Air Quality;
- \( N_{IAQ} \) – Air Quality Number;
- \( ADI \) – Air Distribution Index.

Introduction
This work considers the inlet above the head level, from two rectangular duct, and the outlet also above the head level. Some authors show applications of ceiling mounted localized air distribution systems, such as (Yang et al., 2010) and (Yang et al., 2009). The descendent airflow, promotes by the ceiling mounted localized air distribution system, guarantee an airflow around the occupants to improve the thermal comfort, indoor air quality and Draught Risk levels. The thermal comfort level is evaluated by the Predicted Mean Vote (PMV) index and Predicted Percentage of Dissatisfied (PPD) index, see (Ole Fanger, 1970), (ISO, 2005) and (ASHRAE-55, 2017). These standards determine the thermal comfort conditions in occupied spaces and to define three thermal comfort categories (A, B, C) requirements.

The Carbon Dioxide concentration is used as an indicator to evaluating indoor air quality and ventilation effectiveness. The carbon dioxide concentration, that acceptable limits is defined in the (ASHRAE, 2016) and (Ministério das Obras Públicas, 2006), used as indicator of the indoor air quality (Conceição et al., 2008a), as example. In this work, the same indicator of the indoor air quality was evaluated (Conceição et al., 2008a).

The Draught Risk level, caused by air movement in a human body section, that promotes unwanted local cooling, developed by (Fanger et al., 1988), depends on the temperature, velocity and turbulence intensity of the air, and can be analysed in (ISO, 2005).

In order to evaluate simultaneously the thermal comfort and the air quality, and also the effectiveness for heat removal and the effectiveness for contaminant removal that a ventilation system promotes to the occupants the Air Distribution Index is used. The Air Distribution Index was presented and detailed in the studies of (Awbi, 2004), for uniform environment, and (Conceição et al., 2013), for non-uniform environments, and it allows to evaluate and compare the performance of Heating, Ventilating and Air Conditioning systems.

The numerical simulation considers three software: Computer Fluid Dynamics and Human Thermal Response, that work in a coupling methodology, and the Building Dynamic Response.
The Computer Fluid Dynamics, that simulate the airflow inside the space and around the occupants, and used in this work, can be seen in (Conceição & Lúcio, 2016), as example. The development and validation of this numerical simulation were made in a study of airflow inside office compartments with moderate environments.

The Human Thermal Response, that simulate the human thermal and thermo-physiology systems, was applied in (Conceição, 2000), (Conceição et al., 2010) and (Conceição & Lúcio, 2001), as example. In the first one (Conceição, 2000), the development of the model and a numerical application in thermal comfort and local thermal discomfort were evaluated. In the second one (Conceição et al., 2010), the experimental and numerical application in order to evaluate the thermal comfort level that occupants are subject in desk equipped with personalized ventilations was made. In the third one (Conceição & Lúcio, 2001), the thermal comfort model was validated using numerical and subjective responses of human thermal sensation.

Finally, the Building Dynamic Response, which evaluates the air temperature distribution, surfaces temperature distribution and energy consumption, simulate Building Dynamic Response with complex topology. Application, and more numerical details, were presented using internal energy and comfort performance (Conceição et al., 2008b), using shading devices methodologies (Conceição & Lúcio, 2010b), using buildings with complex topologies (Conceição & Lúcio, 2009), using control models in the Heating, Ventilating and air Conditioning systems control based in human thermo-physiology (Conceição et al., 2018), as example. The Building Dynamic Response numerical model was validated in winter and summer conditions.

The numerical models used in this work are based on a coupling between Computer Fluid Dynamics and Human Thermal Response numerical models, with inputs from the Building Dynamic Response. The validation of an application with simultaneously these three numerical software, as example, using experimental and numerical values of the chamber surface temperature, air temperature, air velocity, air turbulence intensity and Draught Risk around the occupants can be seen in (Conceição & Lúcio, 2016).

The purpose of this work is to develop a new ceiling mounted localized air distribution system built with an inlet system and an outlet system, located above the head level. This numerical simulation considers high occupation spaces with important airflow and heat exchange interaction between occupants. It also considers three software, being one using to evaluate the boundary conditions and the other two using in a coupling methodology. The indoor air quality level, thermal comfort level and Air Distribution Index are evaluated, for winter conditions, in a classroom/auditorium section with high occupation level.

**Numerical Model**

The numerical model, that simulates the Building Dynamic Response, can be analysed, as example, in (Conceição et al., 2000) and (Conceição & Lúcio, 2010a). This numerical software calculated the boundary conditions, namely, the surrounding surface temperature and airflow rate, used as input in the coupling numerical models.

The coupling of the Computer Fluid Dynamics and Human Thermal Response numerical models evaluates the thermal comfort and air quality that the occupants are subjected.

The Human Thermal Response is built by sub-models that simulate the human body thermal system, thermoregulatory system and clothing system. This software calculated the human body temperature, blood flow in the human body, water transpiration flow, blood (arterial and venous) temperature, clothing temperature and water flow through the clothing. The Human Thermal Response numerical model works in transient and steady-state conditions. The Human Thermal Response is built by sub-models, that uses the PPD and PMV indexes, to evaluate the thermal comfort level, considering all heat fluxes verified in the human body. More details can be seen in (Conceição, 2000).

The Computer Fluid Dynamics numerical model, presented in Conceição et al. (2013), that works in steady-state conditions, considers the isothermal thermal conditions and non-isothermal thermal conditions (Conceição & Lúcio, 2016).

In the partial differential equations discretization, used in the Computer Fluid Dynamics numerical model developed by the authors and applied in this work, the finite volume method is used. The hybrid scheme is used in the convective/diffusive fluxes. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used in the velocity and pressure equations. The non-uniform methodology is used in the grid generation. The grid is refined near the surfaces and in the airflow inlet and outlet. The temperature equation considers the density effect neglectable. The vertical air velocity equation considers the impulsion term. The carbon dioxide equation considers the source term in the breathing areas. The iterative TDMA (Tri-Diagonal Matrix Algorithm) method is used in the equations system resolution. The RNG turbulence model, for high Reynolds number, is used in the turbulence simulation. In the wall boundary the surface proximity is used. The Computer Fluid Dynamics numerical model evaluates the air temperature, air velocity, carbon dioxide concentration, turbulent kinetic energy, turbulent energy dissipation, indoor air quality level, Draught Risk level, and others parameters.

The flux used in the numerical simulation, using the three numerical software, are the following:

- In the first step, the Building Dynamic Response is used to evaluate the temperature of the surrounding surfaces in function to the renovation airflow, occupation and outdoor environment;
- In the second step, the Computer Fluid Dynamics calculates the environmental variables around the
occupants, using the body and clothing temperatures (calculated by the Human Thermal Response) as boundary conditions around the manikin;

- In the third step, the Human Thermal Response calculates the body and clothing temperature, using the environmental variables around the manikins (calculated in the Computer Fluid Dynamics) as boundary conditions;
- The iterative method, used sequentially the second and third step, stop when the convergence is acceptable.

The Air Distribution Index, developed in (Awbi, 2004) for uniform environments and adapted in (Conceição et al., 2013) to non-uniform environments, uses the thermal comfort number and the air quality number. The first one considers the effectiveness for heat removal and the PPD, while the second one considers the effectiveness for contaminant removal and the percentage of dissatisfied associated to the indoor air quality.

### Numerical Methodology

The new ceiling mounted localized air distribution systems consider an inlet system, placed above the head level and above the occupation area, and an outlet system, also placed above the head level but not in the occupied area (between the two occupation areas).

This numerical study is made in a virtual chamber, that simulate a classroom/auditorium. The chamber is occupied by twenty-four occupants and equipped with twenty-four seats.

The virtual chamber, equal to an existing experimental chamber, simulates a virtual classroom/auditorium, with a volume of 4.5×2.55×2.5 m³.

The inlet system considers one jet located above the head level of each occupant, while the outlet considers six vertical ducts in the central area of the virtual classroom/auditorium:

- An inlet system built with two horizontal rectangular ducts. Each inlet horizontal rectangular duct considers twenty four jets located above the head level (see Figures 1 to 4);
- An outlet system, built with six vertical outlet ducts. Each extraction duct is divided in a vertical duct, connected to a horizontal duct (see Figures 1 and 2).

In the numerical simulation, made in winter conditions, the inlet air velocity is:

- Case A - 3 m/s (airflow acceptable for 18 occupants);
- Case B - 4 m/s (airflow acceptable for 24 occupants);
- Case C - 5 m/s (airflow acceptable for 30 occupants).

In Figure 1 is presented the scheme of the virtual chamber, equipped with ceiling mounted localized air distribution systems, the occupants, the desks, and ventilation system, using in the Computational Fluids Dynamics.

The scheme of the virtual chamber, equipped with ceiling mounted localized air distribution system, the occupants, the desks, and the ventilated system, used in the Human Thermal Response, is depicted in Figure 2. The grid presented in the Figure 2 is used in the calculation of the heat exchanges by radiation:

- Between the occupants;
- Between the occupants and the surrounding surfaces of the desk and ventilation system.

The grid generation used in the Human Thermal Response is transferred from the Computational Fluids Dynamics numerical model.

Figure 3 shows the location of the occupants and the number identification of the occupants seated in the experimental chamber.

This work, that simulates a classroom/auditorium section, considers four rows of students, namely, two located in the corridor side and two located in the wall/window side.
The inlet airflow location used in the Computational Fluids Dynamics numerical simulation is depicted in Figure 1. The inlet and the outlet are located 1.8 m above the floor level.

The occupants’ clothing level, used in the Human Thermal Response numerical model, in winter conditions, is 1 Clo and the activity level of the occupants is 1.2 Met. The clothing level, considered in this work, is formed by long sleeved shirt, dust, pants, shoes, and normal underwear.

The considered outdoor temperature was 0ºC and the thermal power energy consumption, in order that the mean indoor air temperature is 20ºC, is:

- Case A – 1617 W;
- Case B – 2900 W;
- Case C – 4171 W.

Results and Discussions

In this section the air velocity, the air temperature, the Draught Risk and the Air Distribution Index are presented.

Figures 4, 6 and 8 show, respectively, the air velocity, air temperature and Draught Risk around the occupants. Figures (a), (b) and (c) are associated to, respectively, inlet air velocity of 3, 4 and 5 m/s.

The mean values of air velocity, air temperature and Draught Risk are calculated using all values around each of the 24 occupants’ sections.

Figure 5 shows the air velocity field in a vertical plan located in the occupants placed in the wall/windows side and placed in the corridor side. Figure 7 shows the air temperature field in a vertical plan located in the occupants placed in the wall/windows side and placed in the corridor side.
Figure 5: Air velocity field in a vertical plan located in the occupants placed in the a) wall/windows side and placed in the b) corridor side.

The Draught Risk around the occupant section increases when the inlet air velocity increases. In general, the Draught Risk around the occupant is relatively uniform. However, the occupants located in the corridor side, present higher Draught Risk in the upper member located in the corridor side than in the upper member located in the wall/window side. The Draught Risk level is acceptable in accordance with the category C (30% of dissatisfied people).

The obtained Air Distribution Index value, and all associated parameters, is shown in Tables 1, 2 and 3, respectively, when the inlet air velocity is 3, 4 and 5 m/s.

The carbon dioxide concentration in the respiration area decreases slightly when the inlet air velocity increases and the PPD index increases slightly when the inlet air velocity increases.

The effectiveness for heat removal and the effectiveness for contaminant removal decrease slightly when the inlet air velocity increases.

The Air Distribution Index increases slightly when the inlet air velocity increases. This fact is associated with the air quality number increasing when the inlet air velocity increased; however, the thermal comfort number decreases when the inlet air velocity increases.

Figure 6: Air temperature around the occupants (Tair), when the inlet air velocity is 3 (a), 4 (b) and 5 (c) m/s.
Figure 7: Air temperature field in a vertical plan located in the occupants placed in the a) wall/windows side and placed in the b) corridor side.

Figure 8: Draught Risk around the occupants (DR), when the inlet air velocity is 3 (a), 4 (b) and 5 (c) m/s.

Table 1: Obtained ADI values, when the inlet air velocity is 3 m/s.

<table>
<thead>
<tr>
<th>N</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tm (ºC)</td>
<td>24,3</td>
<td>24,6</td>
<td>24,8</td>
<td>24,8</td>
<td>24,8</td>
<td>24,4</td>
<td>24,5</td>
<td>24,8</td>
<td>24,9</td>
<td>24,9</td>
<td>24,8</td>
<td>24,4</td>
</tr>
<tr>
<td>ETC (%)</td>
<td>70,3</td>
<td>68,4</td>
<td>67,7</td>
<td>67,4</td>
<td>67,0</td>
<td>66,4</td>
<td>65,1</td>
<td>64,6</td>
<td>63,3</td>
<td>62,0</td>
<td>61,2</td>
<td>60,5</td>
</tr>
<tr>
<td>NIAQ</td>
<td>5,7</td>
<td>5,7</td>
<td>5,5</td>
<td>5,4</td>
<td>5,3</td>
<td>5,2</td>
<td>5,0</td>
<td>4,9</td>
<td>4,7</td>
<td>4,6</td>
<td>4,5</td>
<td>4,3</td>
</tr>
<tr>
<td>ADI</td>
<td>8,0</td>
<td>8,0</td>
<td>7,9</td>
<td>7,8</td>
<td>7,7</td>
<td>7,1</td>
<td>7,8</td>
<td>7,8</td>
<td>7,5</td>
<td>7,4</td>
<td>7,4</td>
<td>6,9</td>
</tr>
</tbody>
</table>

Table 2: Obtained ADI values, when the inlet air velocity is 4 m/s.

<table>
<thead>
<tr>
<th>N</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tm (ºC)</td>
<td>24,8</td>
<td>25,0</td>
<td>25,2</td>
<td>25,2</td>
<td>25,2</td>
<td>25,0</td>
<td>25,1</td>
<td>25,2</td>
<td>25,3</td>
<td>25,3</td>
<td>25,2</td>
<td>25,0</td>
</tr>
<tr>
<td>ETC (%)</td>
<td>74,5</td>
<td>73,1</td>
<td>71,9</td>
<td>71,4</td>
<td>71,0</td>
<td>70,3</td>
<td>70,0</td>
<td>69,3</td>
<td>68,6</td>
<td>68,0</td>
<td>66,8</td>
<td>64,8</td>
</tr>
<tr>
<td>NIAQ</td>
<td>4,4</td>
<td>4,3</td>
<td>4,2</td>
<td>4,1</td>
<td>4,0</td>
<td>3,9</td>
<td>3,7</td>
<td>3,5</td>
<td>3,4</td>
<td>3,2</td>
<td>3,1</td>
<td>3,0</td>
</tr>
<tr>
<td>ADI</td>
<td>7,7</td>
<td>7,7</td>
<td>7,5</td>
<td>7,5</td>
<td>7,4</td>
<td>6,9</td>
<td>7,4</td>
<td>7,3</td>
<td>7,1</td>
<td>7,0</td>
<td>7,0</td>
<td>6,6</td>
</tr>
</tbody>
</table>

Table 3: Obtained ADI values, when the inlet air velocity is 5 m/s.

<table>
<thead>
<tr>
<th>N</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tm (ºC)</td>
<td>25,2</td>
<td>25,4</td>
<td>25,6</td>
<td>25,6</td>
<td>25,6</td>
<td>25,4</td>
<td>25,5</td>
<td>25,6</td>
<td>25,7</td>
<td>25,7</td>
<td>25,6</td>
<td>25,4</td>
</tr>
<tr>
<td>ETC (%)</td>
<td>78,5</td>
<td>76,9</td>
<td>75,7</td>
<td>75,2</td>
<td>74,8</td>
<td>74,0</td>
<td>73,7</td>
<td>72,9</td>
<td>72,2</td>
<td>71,5</td>
<td>70,8</td>
<td>69,1</td>
</tr>
<tr>
<td>NIAQ</td>
<td>3,8</td>
<td>3,7</td>
<td>3,6</td>
<td>3,5</td>
<td>3,4</td>
<td>3,3</td>
<td>3,1</td>
<td>3,0</td>
<td>2,9</td>
<td>2,8</td>
<td>2,7</td>
<td>2,6</td>
</tr>
<tr>
<td>ADI</td>
<td>8,5</td>
<td>8,4</td>
<td>8,3</td>
<td>8,2</td>
<td>8,1</td>
<td>7,6</td>
<td>8,2</td>
<td>8,1</td>
<td>7,8</td>
<td>7,7</td>
<td>7,6</td>
<td>7,2</td>
</tr>
</tbody>
</table>
Conclusion

In this work a new ceiling mounted localized air distribution system built with an inlet system and an outlet system, located above the head level is presented. The numerical simulation considers three software. The indoor air quality, thermal comfort and Air Distribution Index are evaluated, for winter conditions, in a classroom/auditorium section with high occupation level. In general, the air velocity, air temperature and Draught Risk around the human body section are relatively constant. However, the occupants located in the corridor side, present higher air velocity and Draught Risk in the upper member located in the corridor side than in the upper member located in the wall/window side.

The air quality number increases when the inlet air velocity increases and the thermal comfort number decreases when the inlet air velocity increases. Thus, the Air Distribution Index increases slightly when the inlet air velocity increases.

However, the thermal comfort level (evaluated by the PPD index) and the indoor air quality are acceptable for all Cases. The first situation, when the inlet air velocity is 3 m/s, Case A, is one acceptable Case, inclusively, with low energy consumption level.

Acknowledgement

The authors would like to acknowledge to the project (SAICT-ALG/39586/2018) from Algarve Regional Operational Program (CRESC Algarve 2020), under the PORTUGAL 2020 Partnership Agreement, through the European Regional Development Fund (ERDF) and the National Science and Technology Foundation (FCT).

References


