Title	Formal calibration methodology for CFD models of naturally ventilated indoor environments
Author(s)	Hajdukiewicz, Magdalena; Geron, Marco; Keane, Marcus M.
Publication Date	2012-08-31
Publication Information	Hajdukiewicz, Magdalena, Geron, Marco, & Keane, Marcus M. (2013). Formal calibration methodology for CFD models of naturally ventilated indoor environments. Building and Environment, 59, 290-302. doi: http://dx.doi.org/10.1016/j.buildenv.2012.08.027
Publisher	Elsevier
Link to publisher's version	http://dx.doi.org/10.1016/j.buildenv.2012.08.027
Item record	http://hdl.handle.net/10379/6177
DOI	http://dx.doi.org/10.1016/j.buildenv.2012.08.027

Formal calibration methodology for CFD models of naturally ventilated indoor environments

Ms. Magdalena Hajdukiewicz

Dr. Marco Geron

Dr. Marcus M. Keane

Informatics Research Unit for Sustainable Engineering (IRUSE)

Department of Civil Engineering and Ryan Institute, National University of Ireland, Galway, Ireland

Corresponding author: *Magdalena Hajdukiewicz*
Department of Civil Engineering and Ryan Institute, National University of Ireland, Galway, Ireland
E-mail: magdalena.hajdukiewicz@nuigalway.ie
Phone: +353 (0) 91 49 2733

Abstract

Well planned natural ventilation strategies and systems in the built environments may provide healthy and comfortable indoor conditions, while contributing to a significant reduction in the energy consumed by buildings. Computational Fluid Dynamics (CFD) is particularly suited for modelling indoor conditions in naturally ventilated spaces, which are difficult to predict using other types of building simulation tools. Hence, accurate and reliable CFD models of naturally ventilated indoor spaces are necessary to support the effective design and operation of indoor environments in buildings.

This paper presents a formal calibration methodology for the development of CFD models of naturally ventilated indoor environments. The methodology explains how to qualitatively and quantitatively verify and validate CFD models, including parametric analysis utilising the response surface technique to support a robust calibration process. The proposed methodology is demonstrated on a naturally ventilated study zone in the library building at the National University of Ireland in Galway. The calibration process is supported by the on-site measurements performed in a normally operating building. The measurement of outdoor weather data provided boundary conditions for the CFD model, while a network of wireless sensors supplied air speeds and air temperatures inside the room for the model calibration.

The concepts and techniques developed here will enhance the process of achieving reliable CFD models that represent indoor spaces and provide new and valuable information for estimating the effect of the boundary conditions on the CFD model results in indoor environments.

Keywords

CFD; indoor environment; natural ventilation; methodology; calibration; parametric analysis

1. Introduction

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

The building sector is responsible for 40% of the total energy consumption in the European Union [1]. Half of the energy consumed by buildings is due to the use of heating, ventilation and air conditioning (HVAC) systems [2]. Recently, the use of natural ventilation as an effective solution to save energy in buildings is actively promoted. Well planned natural ventilation systems provide comfortable and healthy indoor conditions. However, a proper effective design of the naturally ventilated space requires a good understanding of the airflow patterns inside and outside the building.

In recent years CFD has become a very powerful and popular tool in building simulation. CFD has been widely used to model indoor and outdoor airflow, heat transfer and contaminant transport. The applications of CFD in building design include site planning, natural ventilation studies, HVAC system designs or pollution dispersion and control [3]. About 70% of ventilation performance studies published in 2008 used CFD in their analysis [4]. CFD is particularly suited for modelling indoor conditions in naturally ventilated spaces, which are difficult to predict using other types of building simulation tools [4].

There has been extensive research focusing on CFD modelling of natural ventilation in buildings (e.g. [5] - [8]). However, the reliability of the results remains a significant concern regarding CFD simulations. While CFD models can produce visually appealing results, accuracy is often a key issue [9]. In order to produce credible and verifiable results, the CFD model should be created using verified software and experimental data to support model validation [10]. At the same time, the simulation of indoor environment requires the expertise to handle complex boundary conditions [11]. Current research has focused on the topic of verification and validation (V&V) of CFD models in general (e.g. [12] - [15]) and of the built environments (e.g. [16], [17]). In addition, the influence of the CFD simulation input parameters on the results has been analysed. This includes: model geometry (e.g. [18], [19]), turbulence models (e.g. [20], [21]), wind speed (e.g. [22], [23]) and wind direction (e.g. [22], [24]), heat sources (e.g. [25] - [27]) and contaminant sources (e.g. [25]). The comprehensive study from Ramponi and Blocken [28] showed the impact of the size of the computational domain, grid resolution, inlet kinetic energy, turbulence model, discretization scheme and level of convergence on the results of the CFD simulation of cross-ventilation. Furthermore, the external ambient air temperature was found to have a significant effect on air temperature distribution inside a naturally ventilated atrium building [29]. Chourasia and Goswami [30] examined how product and operating conditions influenced the heat and mass transfer within stacked bags of potatoes. While, Zhai and Chen [31] performed sensitivity analysis to determine building and environmental characteristics that could influence the performance of coupling between energy simulation and CFD. Regarding the occupants of the built environments, Topp et al. [32] explored how the size and geometry of human manikin simulated by CFD influenced its personal exposure and CO_2 concentrations.

Despite the presence of best practice guidelines for creating, verifying and validating CFD models, no methodical procedures for the calibration of CFD models of indoor environments have been

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

developed. Calibration is, de facto, the adjustment of numerical and physical model input parameters to amend the agreement between the model results and corresponding experimental data [12]. The motivation for systematically calibrated CFD models of indoor environments is (i) the reliable prediction of the environmental conditions that meets the agreement with the on-site measurements; and (ii) the possibility of the accurate simulation of the environment with varying input parameters. Reliability and robustness of CFD simulations are critical in situations when physical testing is too expensive or infeasible. There is a demand for accurate CFD models of indoor environments at the design and the operation stage of building life cycle [33]. At the same time, true models of the built environments may lead to research in new directions, e.g. support the creation of reduced order models to control indoor environments with a reduction in computational cost, e.g. [34], [35].

2. Objectives

The goal of the research was to develop a consistent and systematic calibration methodology of CFD models of indoor environments in buildings. During the CFD model calibration process the input boundary conditions that most influenced the model output were identified. Those input parameters were adjusted to obtain a reliable CFD model representing the real environment. To date, no systematic calibration methodologies, supported by parametric studies to evaluate the effect of model boundary conditions on the results, have been reported in literature.

This work does not aim to evaluate the influence of the turbulence model on the CFD results. Previous research has explored the topic of turbulence modelling extensively (e.g. [36]). This work focused on the development of clear and systematic calibration methodology for the CFD models of naturally ventilated indoor environments. This was supported by evaluation of the critical boundary conditions on resulting indoor air speeds and air temperatures in the modelled room.

3. Proposed methodology

CFD is a powerful tool for modelling interactions within and between fluids and solids. However, developing reliable CFD models requires a high level of expertise, which in many cases may not be available [37]. When generating a valid model of an indoor space knowledge and CFD experience is essential to:

- decide on the level of detail in the geometry, especially human occupants;
- select the type and resolution of the grid;
- select the turbulence model;
- set the boundary conditions;
- set the numerical techniques (e.g. relaxation factor, discretization scheme, iteration number, etc.);
- estimate the validity of the results.

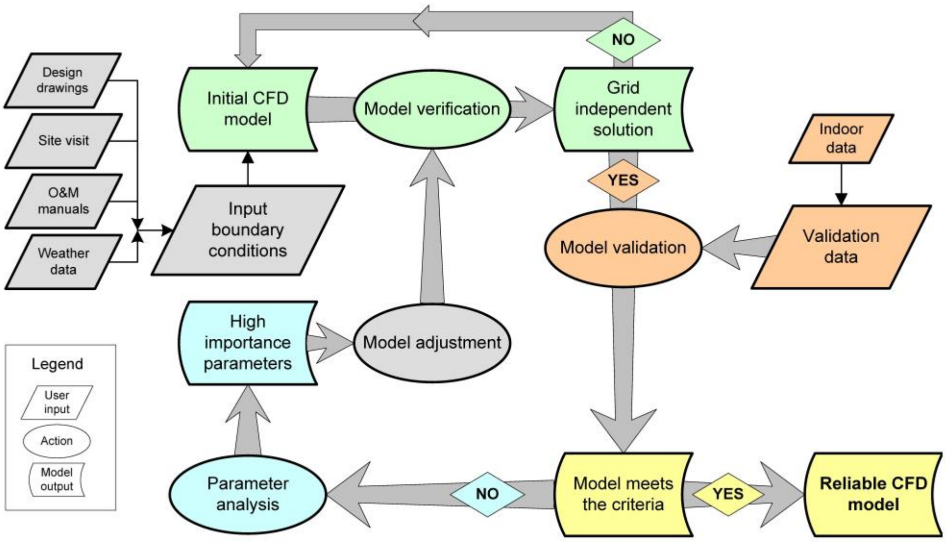
This research presents a formal methodology (Figure 1) that supports the development of reliable calibrated CFD models of naturally ventilated indoor environments. The methodology explains how to

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

verify (green colour) and validate (orange colour) the CFD model, and perform parametric analysis (blue colour) to evaluate the influence of model boundary conditions on the results. The outcome of the calibration procedure is a reliable CFD model that accurately represents the real environment (yellow colour). This research used commercial CFD code Ansys CFX. Ansys CFX is a verified code with many benchmark cases predicting airflow and heat transfer inside indoor spaces. Hence, the work presented here excludes the verification of the CFD code.

Figure 1. Process of achieving a valid CFD model of indoor environment.



Based on the technical documentation, site visits and on-site measurements the initial CFD model was created. Following this, a grid verification study took place. Various runs of the initial CFD model were performed on different size meshes and their results were compared to analyse the grid independence of the solution. A grid independent solution implied the results did not change significantly with increasing number of mesh cells, i.e. the balance between accuracy and computation time was achieved. In this work, the grid independent solution was quantitatively evaluated using the grid convergence index (*GCI*), introduced by Roache [38], for indoor air temperatures.

Once grid independence was established, the simulated air speeds (m/s) and air temperatures (^{o}C) inside the room were validated with on-site measurements. A validation process was performed to

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

prove the ability of the CFD model to predict indoor conditions using available reliable data [17]. The validation criteria depend on the modelled environment, e.g. office spaces require thermal comfort of the occupants, while data centres or clean rooms demand rigorous indoor conditions. This paper describes the calibration methodology based on the example of a study room with an occupant. Thus, the validation criteria were determined on 0.10 m/s and 0.60 ^{o}C absolute differences between measured and simulated air speeds and air temperatures respectively (section 4.3.3). When the model met the specified validation criteria, it was regarded as a true representation of the real environment. If the criteria were not met, a parametric analysis would be performed. Parametric analysis allowed for the determination of the boundary conditions that most influenced the results. The next step was the process of improving the agreement between experimental and simulated data by adjusting the most relevant input parameters. This step was repeated as long as the CFD model met the validation criteria of being a good representation of the real environment. It is worth mentioning, that significant changes in the input parameters might have changed the character of the flow. Thus, the grid independence study should be repeated for the model with new boundary conditions specified.

4. Case study

4.1. Building description

The demonstration building used in this study was a three storey Nursing Library extension to the James Hardiman Library at the National University of Ireland Galway (NUI Galway), Ireland (Figure 2).

Figure 2. Nursing Library building at the National University of Ireland Galway.

The gross floor area of the building was about 800 m^2. The building accommodated library reading spaces, group study rooms, computer study spaces and book stacks. The building was mostly naturally ventilated, except for the air conditioned computer rooms. The internal and external air

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

temperatures, CO_2 concentrations and the energy consumption were monitored by a building management system (BMS). The BMS controlled dampers and window openings depending on the internal air temperatures and CO_2 concentrations. The Nursing Library was opened to the public in September 2009. An easy access to recently created documentation and the building itself allowed for the creation of a CFD model based on the technical drawings, operation and maintenance (O&M) manuals and site visits.

The CFD model of one of the study rooms (Figure 3) on the top floor of the Nursing Library was developed. The dimensions of the room were 2.70 m (D) x 4.46 m (L) x 3.10 m (H). The room was naturally ventilated and heat sources included lights, radiators, computers and human occupants. The external wall, consisting of the windows, allowed for solar irradiation. This wall faced southeast direction with a surface azimuth angle of 66º. The internal wall, opposite to the external wall, contained a glass surface and the door leading to the open reading space. The remaining two internal walls bordered with other study rooms, similar to the one modelled.

Figure 3. Modelled study room in the Nursing Library building.

Please reference as:

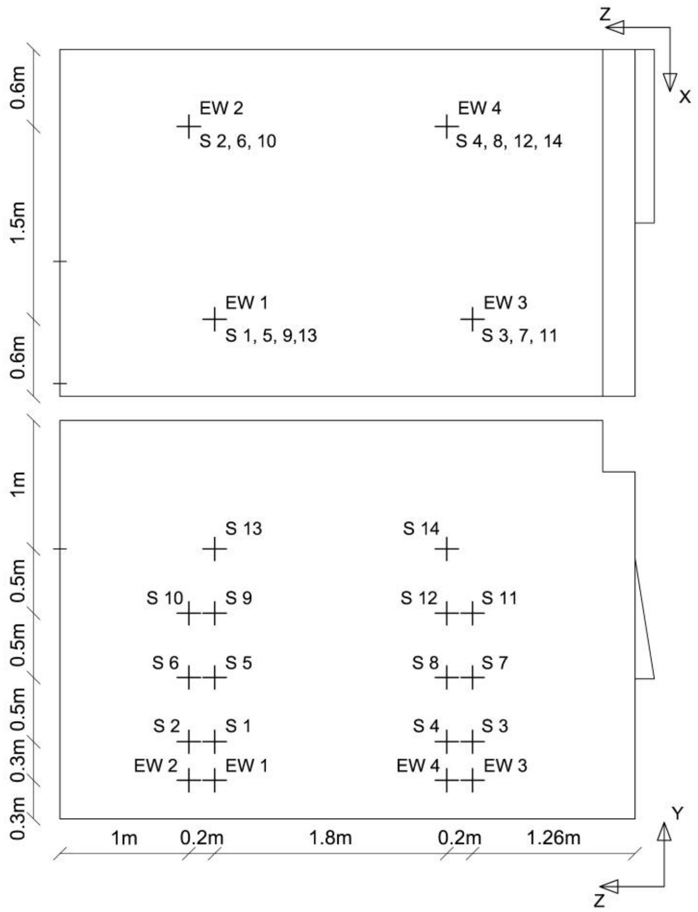
Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

4.2. On-site measurements

A reliable CFD model should be validated with the data obtained from (i) analytical or empirical models; (ii) measurements in small or full-scale laboratory models; or (iii) measurements in the real environment. To support the calibration procedure, this research performed experiments in a normally operating full-scale building exposed to outdoor conditions. The boundary conditions for the CFD model, were provided by the automatic weather station installed at the NUI Galway campus [39] and the air speed sensor placed at the centre of the window opening [40]. To validate the CFD model a network of fourteen wireless air temperature sensors (denoted by the letter S) [40] was deployed. However, after the experiment was carried out, it was discovered that one of the wireless sensors (S8) stopped working during the measurement. In addition, four air speed sensors (denoted by the letters EW) [41] were used (Figure 4). The air temperature sensors were deployed in four horizontal layers to observe the air temperature stratification inside the room. The air speed sensors were located in one horizontal layer near the floor level, where the highest air speeds were expected. All the sensors were deployed in locations where the measurements best described the influence of air speeds and air temperatures on the occupant's thermal comfort.

The air speeds inside the room were measured by the Egg-Whisk wireless sensor network system based on the Tyndall modular prototyping mote [42]. The platform was specifically designed to provide a comprehensive record of the indoor environmental conditions. Within the mote, the airflow sensing was accomplished using a transducer type 54T21 [43] – a self-contained anemometer with a built-in omnidirectional sensor. The air speed sensor was capable to measure indoor convection air speeds between 0.05 – 1 m/s with an accuracy of ± 0.01 m/s. Indoor air temperatures were recorded by the Hobo U12 data loggers [40]. The Hobo U12 data loggers measured the air temperature between -20 ^{o}C and 70 ^{o}C with the accuracy of ± 0.35 ^{o}C in a range between 0 ^{o}C and 50 ^{o}C. The air speed sensor located at the window opening [40] measured the air speed between 0.15 m/s and 5 m/s, with the accuracy greater of 10% of reading or ± 0.05 m/s or 1% full-scale.

Figure 4. The measurement setup.



To gather data that supported CFD model validation, the experiment in the study room was performed on the cloudy day of April 10th, 2011. The external weather conditions, as well as indoor air speeds and air temperatures, were monitored throughout the day. The outdoor air entered the room through the open window. Open internal door allowed for the airflow between the modelled room and adjacent open plan space. The modelled study room was occupied by a sitting human occupant working on a laptop. The conditions over a 12 minute period at noon, when the outdoor and indoor conditions were relatively steady, were chosen to be used in the CFD model. The measurements were taken during the typical operation of the building and maintaining steady conditions inside the room was difficult. It was the authors' intention to take into account the longest continuous period of the experimental steady conditions possible, in order to obtain the plausible ranges of the input parameters used in the parametric study (section 4.5). The average values over the 12 minute period provided the boundary conditions and validation data for the model, while their variations were used in the parametric analysis.

4.3. CFD model

The simulation was performed using the commercial CFD software Ansys CFX v.12.1 [44]. The airflow and the air temperature stratification were simulated in a naturally ventilated room occupied by

Please reference as:

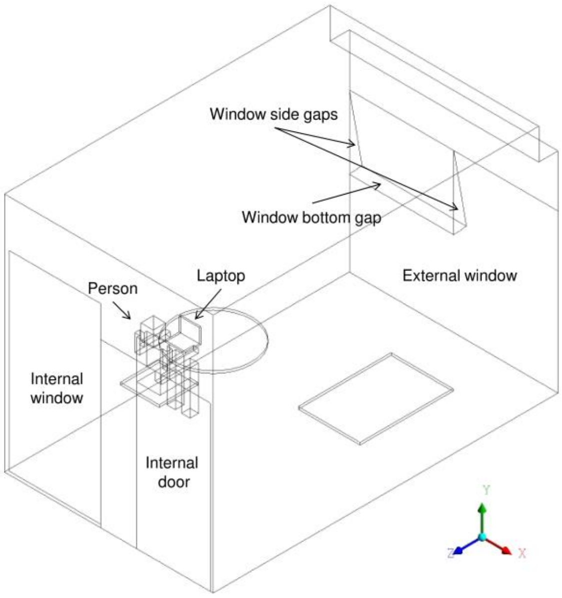
Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

a person working on a laptop. The settings of the CFD model imitated the conditions inside the room during the experiment. The results presented in this section apply to the final model calibrated using proposed methodology. The results of earlier stages of the model development are shown in section 4.6.

4.3.1. Geometry

The 3D geometry of the room was created based on the technical documentation and site visits. Previous research has shown that high level of detail in the CFD model would not influence the overall airflow inside the room but significantly increase grid and computational cost [17]. Additionally, when the focus is on the global airflow of a ventilated space, a simple geometry of a human manikin is sufficient [45]. Thus, the elements of the modelled study room, i.e. occupant, windows, chair, tables, were simplified. Figure 5 shows the level of detail in the geometry of the modelled room.

Figure 5. The geometry of the CFD model.



4.3.2. Model setup

The steady state conditions were used in the CFD analysis of the single phase airflow inside the room. The full buoyancy model, where the fluid density was a function of temperature or pressure, was applied. The air was modelled as ideal gas with the reference buoyancy density of 1.185 kg/m^3 (an approximate value of the domain air density). The standard *k-ε* turbulence model was chosen for good results' accuracy with the robustness of the solution [25], [44]. Satisfactory convergence was achieved using criteria of 0.01% of root mean square residuals for mass and momentum equations, and 1% of the energy conservation target. The boundary conditions of the CFD model are described in Table 1.

The experiment was conducted on a cloudy day. The average total solar irradiance for the period monitored was only 220 W/m^2 [39].The shading outside the windows of the building (Figure 2) blocked a large portion of the radiation coming into the room. Thus, the part of the solar irradiation entering the room was considered not significant in comparison to the heat transfer from the indoor sources. There were two heat sources inside the room - a person and a laptop. The internal walls and the floor were assumed adiabatic. The authors ran two similar CFD simulations to see the influence of the radiation model on the indoor air speeds and air temperatures. Apart from the heat transfer setting that included a person, the boundary conditions for both simulations were the same. In the first simulation, there was no radiation model included and the convective heat flux from the person was set to 60 W/m^2 [17]. In the second simulation, the discrete transfer radiation model (assumption of an isotropic scattering and reasonably homogenous system) was chosen with the person's convective heat flux being set at 18 W/m^2 and a radiative heat flux of 42 W/m^2 (recommended convection to radiation ratio of 30:70 [25]). The results of two models did not differ significantly. However, there were problems with the solution convergence for the second CFD model. For that reason, no radiation model was included in the CFD simulation. Additionally, the surface temperatures were not measured during the experiment due to the lack of available instrumentation. The previous research [25] showed that the additional heat flux of 10W through the room walls influenced the indoor air temperatures and velocities, and improved the agreement between measured and simulated data. In the future work, the authors of the study presented here, will accommodate the need for additional measurements at the room surfaces and incorporation of the radiation model in the CFD simulation.

The outdoor air entered the room through the velocity inlet at the 15 cm wide bottom gap of the window awning. The angle between the boundary plane and the horizontal plane was only 5^o. Because of the relatively small size of the window gap and almost horizontal position of the boundary plane, the air velocity was measured and set in the model as normal to the boundary plane.

Window side gaps (vertical triangular planes formed after opening the window) were modelled as openings at the same relative pressure. The influence of the airflow through the window side gaps was not expected to be significant for the airflow inside the room.

The internal door was specified as the opening to allow for the flow in both directions. The door led to the open reading space, where other windows were open. When measured, there was no pressure difference between the internal door and external conditions; hence, zero relative pressure was specified at the boundary.

Table 1. Boundary conditions for the CFD model.

Boundary	**Type**	**Heat transfer**	**Mass & momentum**
Window bottom gap	Inlet	T_o = 13.38 [^{o}C]	V_{normal} = 0.47 [m/s]
Window side gaps	Opening	T_o = 13.38 [^{o}C]	$P_{relative}$ = 0 [Pa]
Internal door	Opening	T_i = 23.20 [^{o}C]	$P_{relative}$ = 0 [Pa]
External double glazed	Wall	h_c = 2.30 [W/m^{2}K]	No slip wall

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

windows		T_o = 13.38 [ºC]	
Internal single glazed window	Wall	h_c = 4.04 [W/m²K] T_i = 23.2 [ºC]	No slip wall
Internal walls	Wall	Adiabatic	No slip wall
External wall	Wall	Adiabatic	No slip wall
Roof slab/room ceiling	Wall	h_c = 0.15 [W/m²K] T_o = 13.38 [ºC]	No slip wall
Room floor	Wall	Adiabatic	No slip wall
Person sitting	Wall	q = 60 [W/m²]	No slip wall
Laptop	Wall	Q = 30 [W]	No slip wall
Tables & chair	Wall	Adiabatic	No slip wall

** T_o – outdoor temperature; T_i – indoor temperature (for the adjacent reading space); h_c – heat transfer coefficient; q – convective heat flux;Q – convective heat source; V_{normal} – air speed normal to the boundary plane; $P_{relative}$ – pressure relative to the reference pressure ($P_{reference}$ = 102300 Pa - absolute pressure datum from which all other pressure values were taken);*

4.3.3. Grid verification

To verify the CFD model solution the grid convergence study [38] of indoor air temperatures was performed. Three different meshes, successively refined, were created (Table 2) using unstructured elements. The grid refinement ratio *(r)* for 3D mesh was defined as the ratio between the number of grid elements in the fine (Δ_{fine}) and coarse (Δ_{coarse}) meshes:

$$r = \left(\frac{\Delta_{fine}}{\Delta_{coarse}}\right)^{\frac{1}{3}} \quad (1)$$

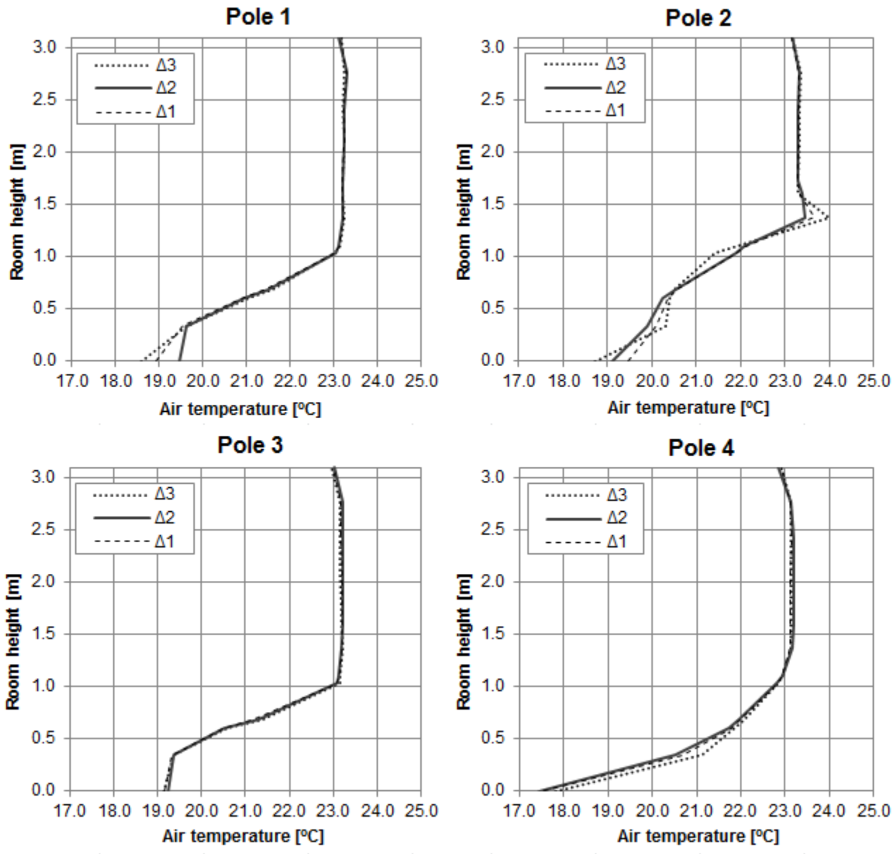
It was recommended to use the grid refinement ratio greater than 1.3 [46] to allow the discretization error to be separated from the other sources of error.

Table 2. Grid parameters for three different mesh sizes.

Δ_3	Δ_2	Δ_1	r_{32}	r_{21}
228 946	581 764	1 532 120	1.36	1.38

Figure 6 represents the qualitative grid verification. Axial air temperature profiles along the room height are plotted for three mesh sizes.

Figure 6. Simulated indoor air temperatures.



The quantitative grid verification was performed using the grid convergence index (*GCI*). Based on the Richardson extrapolation, the *GCI* for the fine grid solution helped to estimate the grid convergence error. The *GCI* was described as:

$$GCI^{fine} = F_s \frac{\varepsilon}{r^p - 1} \tag{2}$$

where F_s was the safety factor equal to 1.25 when comparing three or more grids [46], and p was the order of convergence, calculated from:

$$p = \frac{\ln\left|\frac{f_{coarse} - f_{medium}}{f_{medium} - f_{fine}}\right|}{\ln r} \tag{3}$$

In this analysis, the order of convergence was taken as the average value over the monitored points (p = 1.3).

The *GCI* was evaluated using a relative error between the coarse (f_{coarse}) and fine (f_{fine}) grid solutions:

$$\varepsilon = \frac{f_{coarse} - f_{fine}}{f_{fine}} \tag{4}$$

The *GCI* parameter may be applied to any variable in a CFD solution. In this work the *GCI* was calculated for the air temperatures at 14 sensors' locations inside the modelled room. Those were compared to the measurement accuracy at the same locations to justify the choice of the grid independent solution with the reasonable computation cost (Table 3).

Table 3. Grid convergence index and measurement accuracy for indoor air temperatures.

Sensor location	ε_{32} [10^{-2}]	ε_{21} [10^{-2}]	GCI^{coarse}_{32} [%]	GCI^{fine}_{21} [%]	Measurement accuracy [%]
S1	0.54	0.29	1.32	0.70	1.58
S2	0.78	- 0.55	1.92	- 1.32	1.56
S3	0.26	0.24	0.64	0.58	1.57
S4	0.65	- 0.20	1.59	- 0.49	1.58
S5	0.06	0.05	0.14	0.13	1.55
S6	- 0.66	- 0.15	- 1.62	- 0.36	1.55
S7	0.16	- 0.01	0.39	- 0.03	1.54
S8	- 0.02	0.02	- 0.04	0.04	-
S9	0.02	0.02	0.05	0.04	1.53
S10	- 0.46	0.14	- 1.13	0.33	1.53
S11	- 0.06	0.00	- 0.14	0.00	1.53
S12	- 0.34	0.31	- 0.82	0.75	1.54
S13	0.03	- 0.03	0.07	- 0.06	1.53
S14	- 0.29	0.27	- 0.71	0.66	1.54

The qualitative comparison reported in Figure 6 showed a very good agreement between the medium (Δ_2) and fine (Δ_1) mesh. The small values of the *GCI* confirmed the grid independence of the solution and deemed the grid Δ_2 as the grid used for further analysis. The calculated values of the GCI^{coarse}_{32} (that can be seen as the error estimators associated with the grid resolution) were close to the measurement accuracy, while values of the GCI^{fine}_{21} were unnecessarily low. The grid Δ_2 was thus individuated as the mesh with the right compromise between the level of uncertainty and computational cost.

4.3.4. CFD model results

Table 4 lists the properties of the simulated flow inside the room. The Reynolds number confirmed the flow inside the room was of turbulent nature. With the expected range of indoor air temperatures the dynamic viscosity, thermal conductivity and the coefficient of air thermal expansion were assumed constant.

Table 4. Properties of the flow inside the modelled room.

Reynolds number	44 161 [-]
Prandtl number	0.70462 [-]
Dynamic viscosity	$1.831e^{-5}$ [kg/(ms)]
Thermal conductivity	$2.61e^{-2}$ [W/(mK)]
Thermal expansion coefficient for air	$3.398e^{-3}$ [1/K]

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

Figure 7 shows the locations of three vertical planes: A (through the occupant), B (through the locations EW2, EW4, S2, S4, S6, S8, S10, S12, S14) and C (through the locations EW1, EW3, S1, S3, S5, S7, S9, S11, S13) used to show the CFD model results. Figure 8 represents air temperature stratification and airflow distribution inside the room.

The results showed a stratified fluid inside the room, with a main flow of cold air entering through the window inlet and exiting through the lower part of the door opening. At the same time, the warm air from the open plan space entered the room through the upper part of the door opening (plane C). In planes A and B it was possible to observe the heat plumes generated above the person and laptop heat sources. However, the draught dominated the flow inside the room, so the heat plumes were not as large as expected in a still environment.

The average air speeds inside the room ranged from 0.27 m/s at the ankles level (h = 0.1 m), 0.12 m/s at the sitting person's waist level (h = 0.6 m), to 0.16 m/s at the sitting or standing person's head level ($h_{sitting}$ = 1.1 m; $h_{standing}$ = 1.7 m). Respectively, the average air temperatures were 18.62 ^{o}C at the ankles level, 20.72 ^{o}C at the sitting person's waist level, 21.79 ^{o}C at the sitting person's head level and 22.95 ^{o}C at the standing person's head level.

Figure 7. Location of the vertical planes.

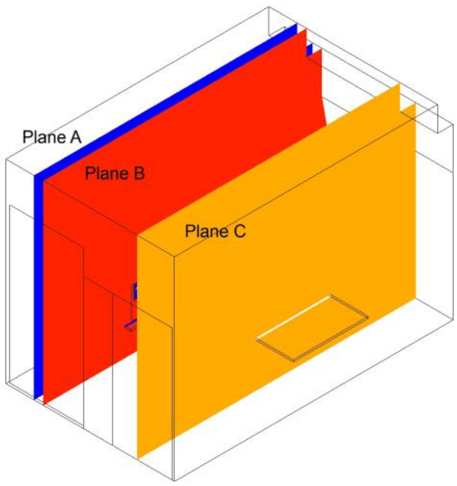
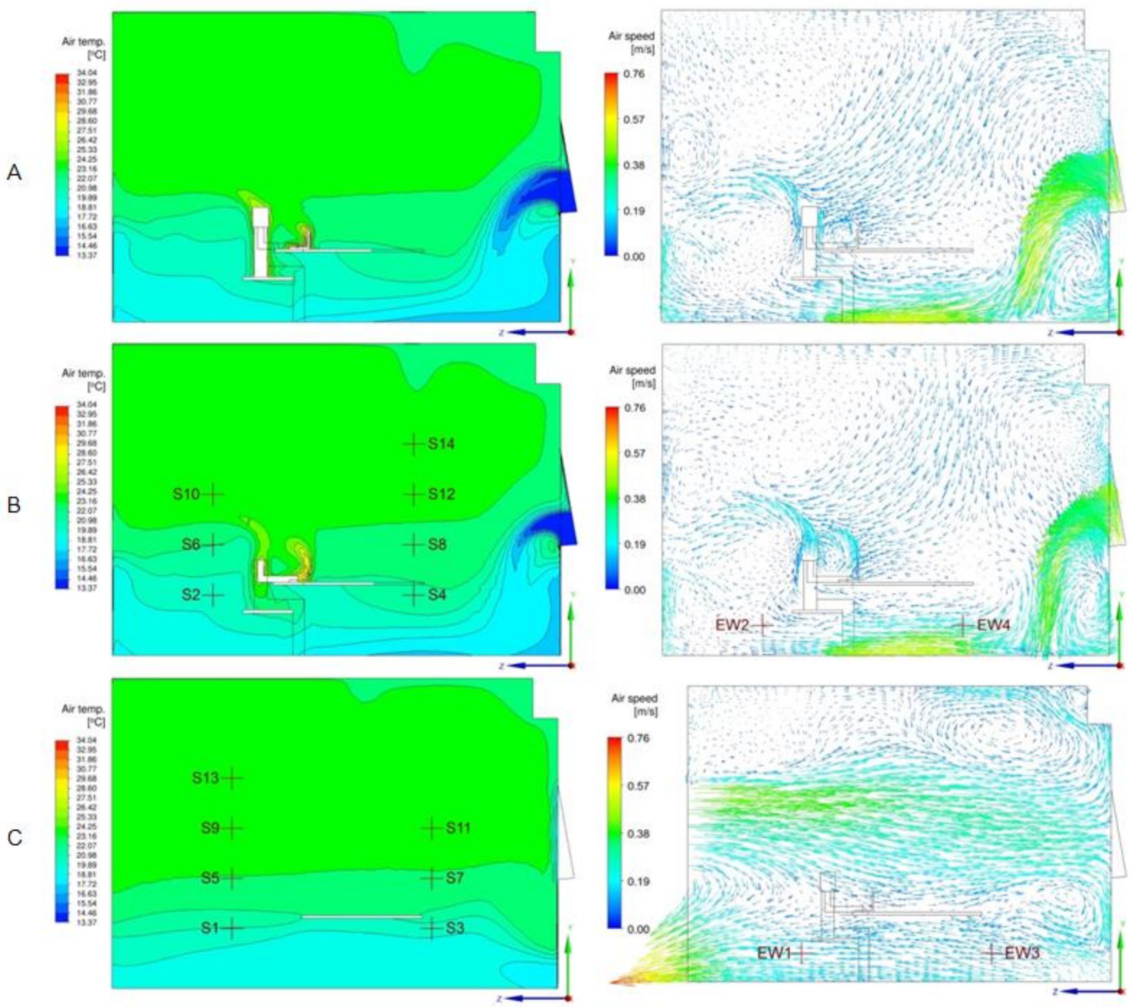


Figure 8. Verified model results.

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027



4.4 CFD model validation

An important step of the calibration procedure is validation of verified CFD model results. Figures 9 & 10 illustrate the comparison between simulated and measured indoor air speeds and air temperatures. Error bars represent the measurement uncertainty (mean value ± standard deviation), including sensors accuracy (± 0.01 m/s for air speeds and ± 0.35 °C for air temperatures). The values of the standard deviations for indoor air speeds ranged between 0.07 - 0.10 m/s, and for air temperatures between 0.08 – 0.22 °C. The graphed results showed a close prediction of the indoor environment.

Office spaces require the calculation of air speeds and air temperatures to evaluate the thermal comfort of the occupants. Based on that, the validation criteria were defined as 0.10 m/s absolute difference between measured and simulated air speeds; and 0.60 °C absolute difference between measured and simulated air temperatures. Those criteria were based on the maximum uncertainties in measured data (0.11 m/s for air speeds and 0.57 °C). Additionally, this air speed criterion was

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

supported by the strictest design specification of the air velocity in buildings (the mean air velocity in single office or kindergarten during the winter season may not exceed 0.10 m/s) [47]. The criterion for the air temperatures was supported by the allowable peak-to-peak variation in indoor air temperatures, which, if less than 1K, causes no influence on the occupants' comfort [47]. Also, the maximum indoor operative temperature change allowed in a 15 minute period is 1.1 °C [48].

Figure 9. Qualitative comparison of measured (range of data shown) and simulated indoor air speeds.

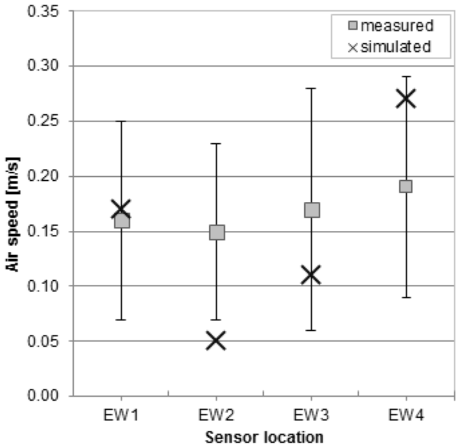
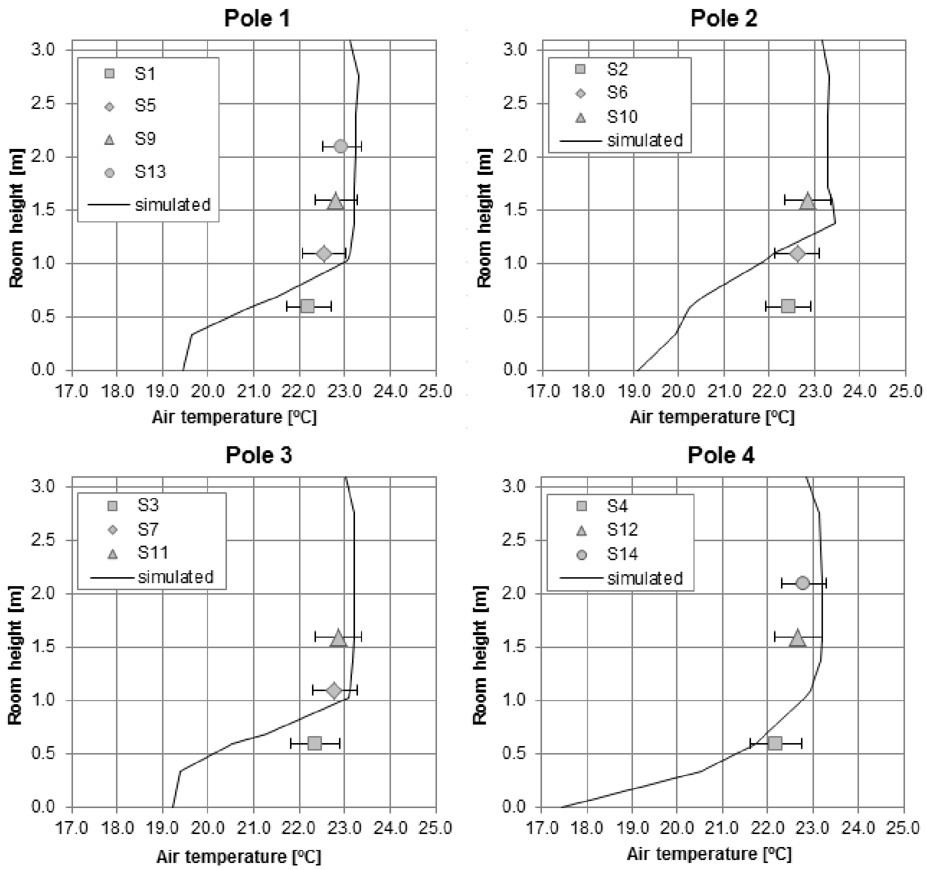


Figure 10. Qualitative comparison of measured (range of data shown) and simulated indoor air temperatures.



Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

Table 5 shows the absolute difference between measured and simulated indoor air speeds and air temperatures. To a significant extent, the model met the validation criteria, and thus provided a good prediction of the indoor airflow and air temperature stratification. This made the CFD model a reliable representation of the environment. At the upper levels of sensor locations inside the room, the air temperatures were slightly overestimated by the model. On the other hand, the model underestimated the air temperatures at the lower level (*h* = 0.6 m). At the locations S1, S2 and S3 the absolute difference in air temperatures exceeded 0.60 °C. A wide range of computational parameters, either numerical or non-numerical (e.g. the assumption of adiabatic room surfaces or turbulence model), might have influenced the discrepancy in CFD predictions. Additionally, the higher air temperature difference at the location S2 might have been caused by the placement of the sensor behind the sitting person. The simplified geometry of the manikin in the model, by blocking the air coming from the window inlet, might have created slightly different flow than in the experiment. The higher difference in air temperatures at the locations S1 and S3 could have been explained by the influence of the airflow at the internal door boundary, which was not known from the experiment and fully predicted by the model. However, further analysis of the other model input parameters (not taken into account in this work), would be required to determine the reasons of the discrepancy between measured and simulated data at the S1, S2 and S3 locations.

Table 5. Quantitative comparison of measured and simulated indoor data.

Data type	Location	Measured	Simulated	Absolute difference
Air speed [m/s]	EW1	0.16	0.17	0.01
	EW2	0.15	0.05	0.10
	EW3	0.17	0.11	0.06
	EW4	0.19	0.27	0.08
Air temperature [°C]	S1	22.21	20.94	**1.27**
	S2	22.42	20.25	**2.17**
	S3	22.36	20.52	**1.84**
	S4	22.19	21.73	0.46
	S5	22.54	23.12	0.58
	S6	22.61	22.09	0.52
	S7	22.78	23.13	0.35
	S9	22.81	23.21	0.40
	S10	22.85	23.38	0.53
	S11	22.87	23.20	0.33
	S12	22.67	23.20	0.53
	S13	22.93	23.23	0.30
	S14	22.80	23.19	0.39

4.5 Parametric analysis

The accuracy of the CFD simulation strongly depends on the model boundary conditions specified by the modeller. It is important to simulate the environment as close as possible to the real conditions. However, detailed information about the boundary conditions is not always easily available for the particular space. Gathering this information may be time consuming. At the same time, some parameters input to the model would more or less influence the results. It is up to a modeller to decide the accuracy of the model input parameters, which may be difficult, especially for inexperienced CFD modellers.

The aim of a parametric analysis showed here, is to determine the importance of input boundary conditions in terms of their contribution to the change in the model output indoor air speeds and air temperatures. The analysis presented here used the response surface methodology [49]. The response surface is a technique to analyse and optimise the response of output parameters influenced by varying input parameters. The response surface can be of the first or second order. When the response surface has a parabolic curvature, a second order model should be used, which is the case in this work. The second order response surface model can be described by regression model:

$$y = \beta_0 + \sum_{j=1}^{q} \beta_j x_j + \sum_{i=1}^{q} \beta_{jj} x_j^2 + \sum\sum_{i<j} \beta_{ij} x_i x_j + \varepsilon \tag{5}$$

where the coefficient β_j measures the expected change in response *y* per unit increase in x_i input parameter. The *i* is the number of observation, *j* the level of independent input variable, ε is a random error and $(i = 1, 2, \ldots, N), (j = 1, 2, \ldots, q), x_i = (x_{1i},\ x_{2i}, \ldots,\ x_{iq}), \beta = (\beta_1,\ \beta_2, \ldots,\ \beta_q)$.

The determination of the values for the input parameters to generate the second order response surface model was performed using the central composite design method with fractional factorial design. Central composite is a design method available for fitting a second order model and consists of three types of points: axial, cube and centre point.

At the first step of the analysis, the plausible ranges for input parameters were assessed (Table 6). Those ranges were related to the variations in the measurements during the experiment and uncertainty in the material properties of the building components. The range of the air reference density (*RefDen*) was calculated using the ideal gas law equation based on the minimum and maximum expected indoor air temperatures. The ranges of the inlet air temperature (*AirTemp*) and inlet air speed (*AirVel*) were based on the minimum and maximum measured air temperature (weather station) and air speed (window inlet). The heat flux generated from the person (*PerHeat*) was taken as minimum for the seated person reading and maximum for the seated person filing [17]. The range of the heat generated by the laptop (*CompHeat*) was based on the manufacturer's specification ± 10%. The ranges of the windows' heat transfer coefficients (*IntGlassHTC, WinHTC*) were calculated based on the uncertainties in thermal resistance values for windows. Finally, the range of the ceiling's heat transfer coefficient (*CeilingHTC*) was calculated based on the structural drawings including the uncertainties in the thickness of roof layers and material properties.

Table 6. Input parameters and their ranges.

Input parameter	Description	Lower bound	Upper bound
RefDen	Air density	1.185 [kg/m^3]	1.222 [kg/m^3]
AirTemp	Outdoor air temperature	13.16 [^{o}C]	13.50 [^{o}C]
AirVel	Air speed at the inlet	0.21 [m/s]	1.80 [m/s]
PerHeat	Person heat flux	55 [W/m^2]	70 [W/m^2]
CompHeat	Laptop heat source	24 [W]	36 [W]
IntGlassHTC	Heat transfer coefficient of the internal window	3.77 [W/m^{2}K]	4.30 [W/m^{2}K]
WinHTC	Heat transfer coefficient of the external window	1.55 [W/m^{2}K]	3.05 [W/m^{2}K]
CeilingHTC	Heat transfer coefficient of the ceiling	0.13 [W/m^{2}K]	0.17 [W/m^{2}K]

For selected eight input parameters, central composite design method generated 81 sets of input parameters, which provided boundary conditions for 81 CFD models to be solved. Changing the input boundary conditions within their ranges caused changes in the output indoor air speeds up to 0.26 m/s and indoor air temperatures up to 9.08 ^{o}C. Based on the 81 sets of input boundary conditions, together with the relevant output indoor air speeds and air temperatures, the response surfaces were created using MATLAB [50].

The accuracy of the fit of the predicted response values in the response values, generated by 81 CFD models, can be estimated using the value of the coefficient of determination R^2 (Table 7). The value of R^2 closer to 1, better the estimation of the regression equation fits the data generated by CFD. The R^2 values for indoor air speeds and air temperatures indicated good accuracy of the responses' prediction.

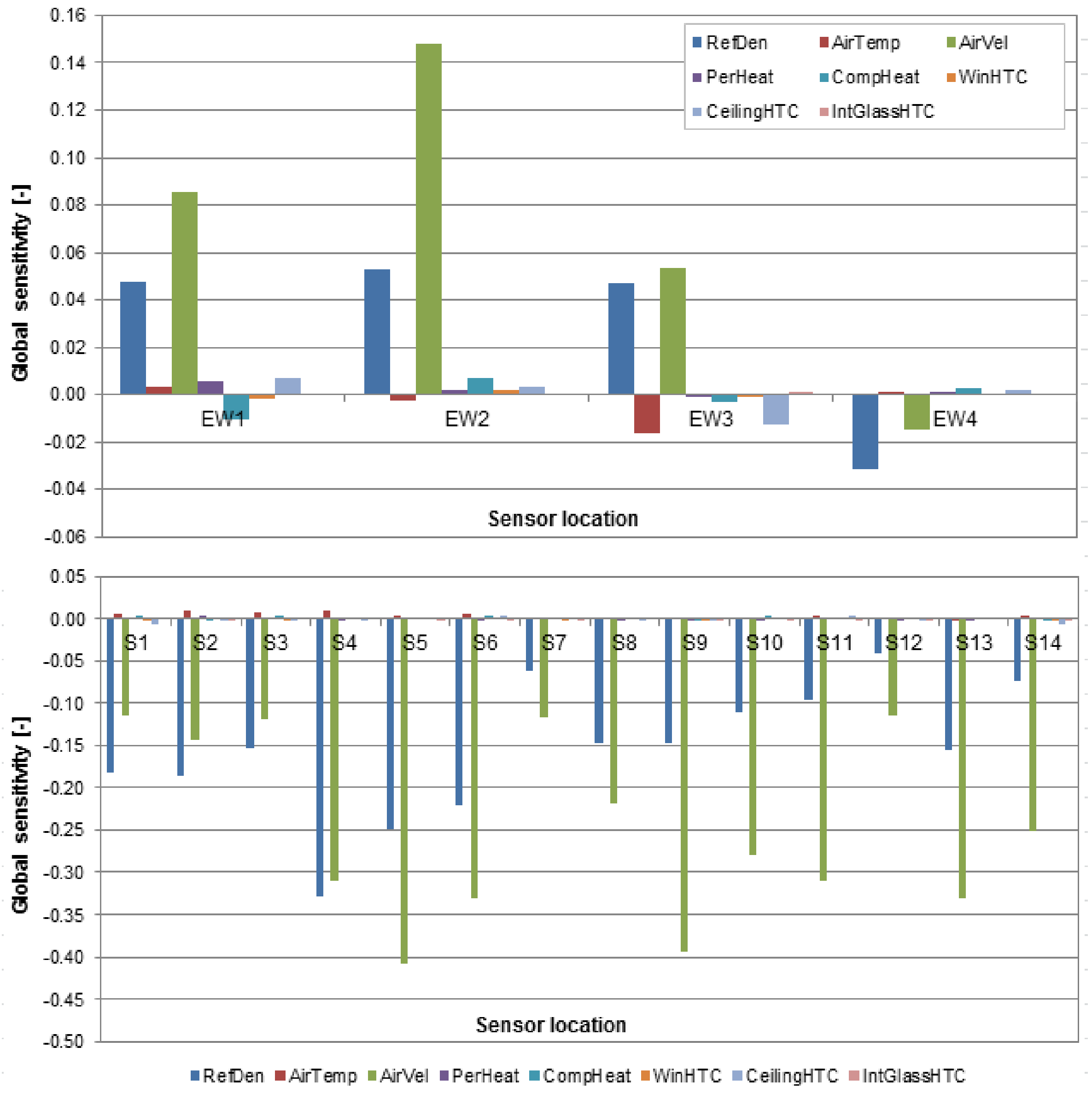
Table 7. Coefficient of determination for simulated indoor air speeds and air temperatures.

Measurement	Sensor location	R^2
Air speed [m/s]	EW1	0.941
	EW2	0.975
	EW3	0.859
	EW4	0.728
Air temperature [^{o}C]	S1	0.884
	S2	0.921
	S3	0.890
	S4	0.978
	S5	0.998
	S6	0.989
	S7	0.987
	S8	0.995

	S9	0.997
	S10	0.998
	S11	0.994
	S12	0.994
	S13	0.998
	S14	0.992

Figure 11 graphs the global sensitivities of the input boundary conditions to output air speeds and air temperatures inside the room. The global sensitivities were calculated based on the coefficient *β*, which was the measurement of the expected change in the output parameters with varying inputs. The input air reference density (*RefDen*) and inlet air speed (*AirVel*) influenced the air temperatures and air speeds inside the room the most. The inlet air temperature (*AirTemp*) had a slight effect on the results; however its variation in the parametric analysis was not large enough to cause a significant change in the indoor conditions. The remaining input parameters, which varied within specified ranges, did not show any significant effect on the relevant outputs.

Figure 11. Sensitivity of input boundary conditions to indoor air speeds and air temperatures.



The response surface plots are presented in order to better visualise the response of the indoor air speeds to the most important boundary conditions, i.e. inlet air speed and modelled air reference density (Figure 12). The plots show minimum, maximum, ridge or saddle point in the response data and may be very useful when analysing the influence of the input boundary conditions to the output results, and optimising the CFD model to better fit the measurement data. Figure 13 displays the impact of inlet air speed and air reference density on indoor air temperatures at the locations S1, S2 and S3 (where the simulated data did not meet the validation criteria). The graphs show only small changes in the air temperatures with changing boundary conditions. This proves the statement inserted in section 4.4, explaining the possible reasons for the absolute difference in air temperatures at S1, S2 and S3 locations exceeding the validation criterion. Further analysis of the additional model input parameters, would be required to determine the reasons of the discrepancies at the S1, S2 and S3 locations.

Figure 12. Indoor air speed responses to changing input air speed and air reference density.

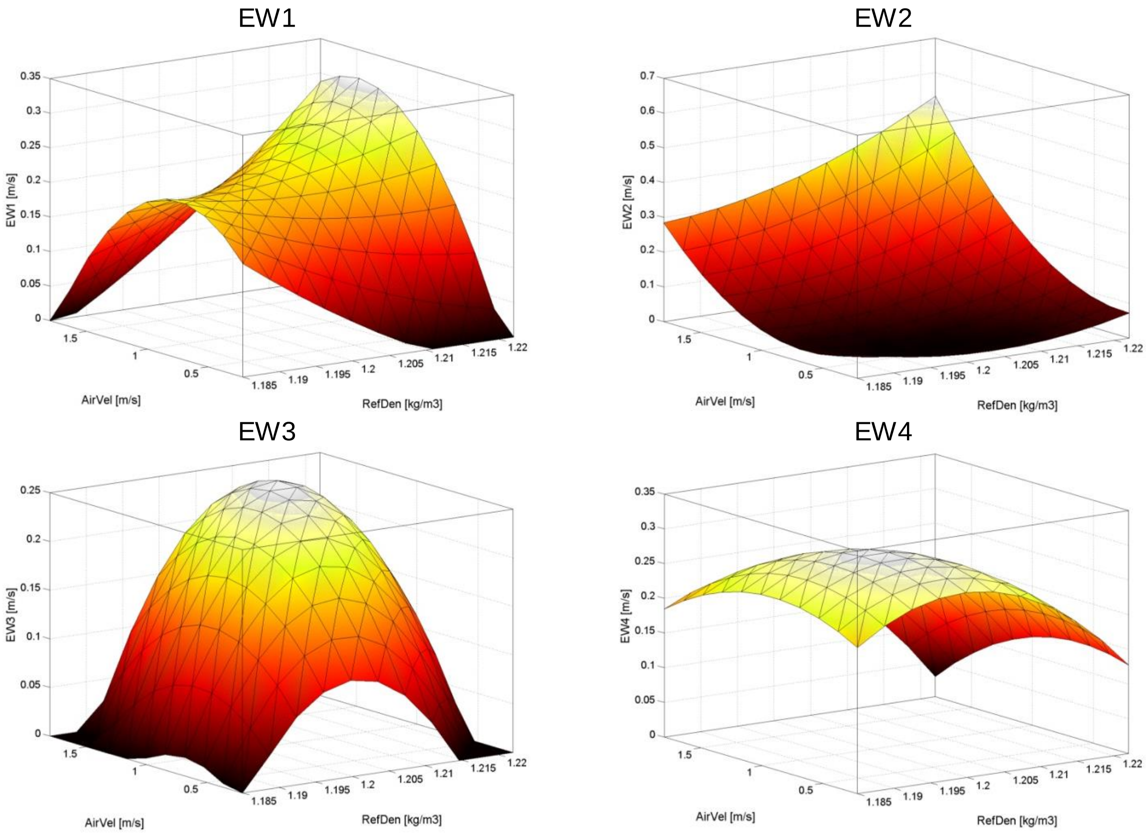
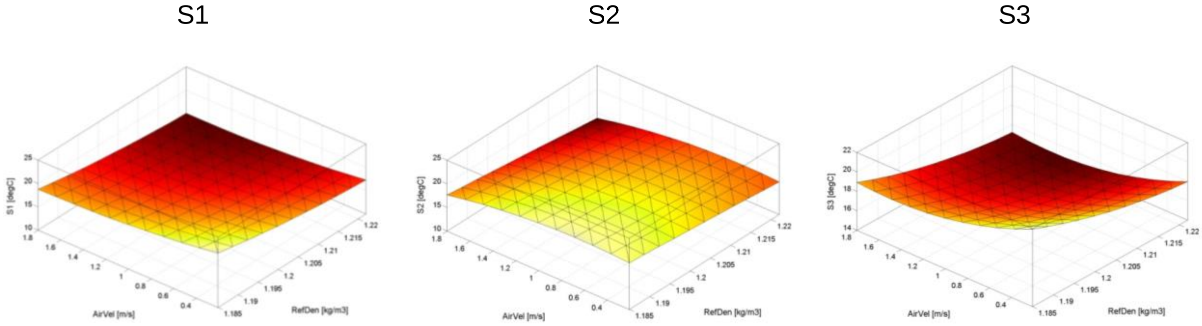


Figure 13. Indoor air temperature responses to changing input air speed and air reference density.



4.6 Evaluation of model improvement

The CFD model described in section 4.3 and validated with the on-site measurements in section 4.4 is the final CFD model obtained by following the steps of the proposed calibration methodology

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

(section 3). Table 8 compares the absolute differences between measured and simulated data for the final and two previous CFD models.

Table 8. Comparison of CFD models generated during the calibration process.

Sensor location	**Absolute difference |measured – simulated|**		
	1st model	**2nd model**	**Final model**
EW1 [m/s]	0.05	0.03	0.01
EW2 [m/s]	0.08	**0.11**	0.10
EW3 [m/s]	0.08	0.06	0.06
EW4 [m/s]	**0.13**	0.02	0.08
S1 [ºC]	**4.85**	**1.12**	**1.27**
S2 [ºC]	**5.10**	**2.01**	**2.17**
S3 [ºC]	**5.32**	**1.84**	**1.84**
S4 [ºC]	**4.36**	0.35	0.46
S5 [ºC]	**2.15**	0.52	0.58
S6 [ºC]	**2.24**	0.28	0.52
S7 [ºC]	**2.16**	0.36	0.35
S9 [ºC]	**0.64**	0.39	0.40
S10 [ºC]	**0.69**	**0.75**	0.53
S11 [ºC]	**0.72**	0.37	0.33
S12 [ºC]	0.52	**0.66**	0.53
S13 [ºC]	0.50	0.38	0.30
S14 [ºC]	0.31	0.56	0.39

First, the 1st model was created with the air reference density set to 1.214 kg/m^3. This caused a significant underestimation of the air temperatures inside the room (S1 – S11). Thus, the air reference density was improved to the value of 1.185 kg/m^3 and the 2nd model was developed. However, this model also showed some discrepancies between measured and simulated data. The model boundary conditions were examined and it was discovered that the heat transfer coefficients (for the ceiling, external and internal windows) were not specified accurately. The changes were made to represent those boundaries truly and the final model was created (which results are presented in this paper). There was a slight improvement in the prediction of air speed at EW2 and air temperatures at S10 and S12. The prediction of air temperatures at S1, S2 and S3 still did not meet the validation criteria. CFD simulations may be sensitive to a wide range of computational parameters [28]. The discrepancy between measured and simulated air temperatures (S1, S2 and S3) might have been caused by

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

inaccurate or wrong assumptions in the model (e.g. assumption of the adiabatic walls and floor or turbulence model), so changing the boundary conditions within their ranges (Table 6) did not improve the prediction at those locations. As mentioned before, further analysis of the model input parameters not taken into account in this work, would be required to determine the reasons of those discrepancies.

5. Conclusions

Well planned natural ventilation strategies in the built environments can provide healthy and comfortable indoor conditions, while contributing to a significant reduction in the energy consumed by buildings. Hence, accurate and reliable CFD models of naturally ventilated indoor spaces are necessary to correctly design and operate built environments. However, creating accurate CFD models, that truly represent real environments, requires a high level of expertise and modeller's time. Furthermore it is widely reported, in the CFD community, that the results of the CFD simulations highly depend on the model parameters input by the user.

This paper describes a formal calibration methodology for the development of CFD models of naturally ventilated indoor environments. Presented methodology, that recognises the practical modelling of occupied spaces in real life environments, is demonstrated on a naturally ventilated study room occupied by a person working on a laptop. The methodology explains how to qualitatively and quantitatively verify and validate CFD model; as well as, perform parametric analysis to support a robust calibration process. Applying the methodology optimised the decision making process when creating a final valid CFD model of the study room. The estimation of the numerical boundary conditions that most influenced model results was successfully facilitated by the response surface method. The concepts and techniques developed here will enhance the process of achieving reliable CFD models that represent indoor spaces and provide new and valuable information for estimating the effect of the numerical boundary conditions on the CFD model results in indoor environments.

Acknowledgements

The authors thank Higher Education Authority for the financial support of this research as part of the NEMBES project. The NEMBES project allows for collaboration between the following institutions: Cork Institute of Technology, University College Cork, National University of Ireland Galway, Trinity College Dublin and University College Dublin. The authors would also like to acknowledge the support of Science Foundation Ireland in funding the National Access Program at the Tyndall National Institute and Enterprise Ireland, which have funded aspects of this work.

References

[1] Official website of the European Union, 2012. [Online]. Available: http://www.europa.eu.

[2] L. Pérez-Lombard, J. Ortiz, and C. Pout, 'A review on buildings energy consumption information', *Energy and Buildings*, vol. 40, no. 3, pp. 394–398, 2008.

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

[3] Z. Zhai, ‘Application of Computational Fluid Dynamics in building design: Aspects and trends’, *Indoor and Built Environment*, vol. 15, no. 4, pp. 305–313, 2006.

[4] Q. Chen, ‘Ventilation performance prediction for buildings: A method overview and recent applications’, *Building and Environment*, vol. 44, no. 4, pp. 848–858, 2009.

[5] S. S. Ayad, ‘Computational study of natural ventilation’, *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 82, no. 1–3, pp. 49–68, 1999.

[6] Y. Jiang and Q. Chen, ‘Study of natural ventilation in buildings by large eddy simulation’, *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 89, no. 13, pp. 1155–1178, 2001.

[7] C. Allocca, Q. Chen, and L. R. Glicksman, ‘Design analysis of single-sided natural ventilation’, *Energy and Buildings*, vol. 35, no. 8, pp. 785–795, 2003.

[8] G. Evola and V. Popov, ‘Computational analysis of wind driven natural ventilation in buildings’, *Energy and Buildings*, vol. 38, no. 5, pp. 491–501, 2006.

[9] K. G. Brill, ‘Don’t be fooled by pretty pictures’, *Forbes*, 2008. [Online]. Available: http://www.forbes.com/.

[10] W. L. Oberkampf and T. G. Trucano, ‘Verification and validation benchmarks’, *Nuclear Engineering and Design*, vol. 238, no. 3, pp. 716–743, 2008.

[11] Q. Chen and Z. Zhai, ‘The use of Computational Fluid Dynamics tools for indoor environmental design’, in *Advanced building simulation*, vol. Chapter 5, Taylor & Francis, 2003, pp. 119–140.

[12] AIAA G-077, *Guide for the verification and validation of Computational Fluid Dynamics simulations*. American Institute of Aeronautics and Astronautics, 1998.

[13] W. L. Oberkampf and T. G. Trucano, ‘Verification and validation in computational fluid dynamics’, *Progress in Aerospace Sciences*, vol. 38, no. 3, pp. 209–272, 2002.

[14] ASME, ‘Guide for verification and validation in Computational Solid Mechanics’, The American Society of Mechanical Engineers, ASME V&V 10-2006, 2006.

[15] D. Pelletier, ‘Verification, validation, and uncertainty in Computational Fluids Dynamics’, *Canadian Journal of Civil Engineering*, vol. 37, no. 7, pp. 1003–1013, 2010.

[16] Q. Chen and J. Srebric, ‘A procedure for verification, validation, and reporting of indoor environment CFD analyses’, *HVAC & R Research*, vol. 8 (2), pp. 201 – 216, 2002.

[17] ASHRAE, *ASHRAE Handbook - Fundamentals*. Atlanta, USA: American Society of Heating, Refrigerating and Air-Conditioning Engineers, 2005.

[18] H. Lee and H. B. Awbi, ‘Effect of internal partitioning on indoor air quality of rooms with mixing ventilation--basic study’, *Building and Environment*, vol. 39, no. 2, pp. 127–141, 2004.

[19] K. Visagavel and P. S. S. Srinivasan, ‘Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening using CFD’, *Solar Energy*, vol. 83, no. 1, pp. 2–5, 2009.

[20] Z. Zhang, W. Zhang, Z. Zhai, and Q. Chen, ‘Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2: Comparison with experimental data from literature’, *HVAC & R Research*, vol. 13, no. 6, pp. 871–886, 2007.

[21] S. Hussain, P. H. Oosthuizen, and A. Kalendar, ‘Evaluation of various turbulence models for the prediction of the airflow and temperature distributions in atria’, *Energy and Buildings*, vol. 48, no. 0, pp. 18–28, 2012.

[22] J. M. Horan and D. P. Finn, ‘Sensitivity of air change rates in a naturally ventilated atrium space subject to variations in external wind speed and direction’, *Energy and Buildings*, vol. 40, no. 8, pp. 1577–1585, 2008.

[23] T. S. Larsen, N. Nikolopoulos, A. Nikolopoulos, G. Strotos, and K.-S. Nikas, ‘Characterization and prediction of the volume flow rate aerating a cross ventilated building by means of experimental techniques and numerical approaches’, *Energy and Buildings*, vol. 43, no. 6, pp. 1371–1381, 2011.

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. ‘Formal calibration methodology for CFD models of naturally ventilated indoor environments’. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

[24] T. van Hooff and B. Blocken, ‘On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium’, *Computers & Fluids*, vol. 39, no. 7, pp. 1146–1155, 2010.

[25] J. Srebric, V. Vukovic, G. He, and X. Yang, ‘CFD boundary conditions for contaminant dispersion, heat transfer and airflow simulations around human occupants in indoor environments’, *Building and Environment*, vol. 43, no. 3, pp. 294–303, 2008.

[26] S. Gendelis and A. Jakovičs, ‘Numerical modelling of airflow and temperature distribution in a living room with different heat exchange conditions’, *Latvian Journal of Physics and Technical Sciences*, vol. 4, 2010.

[27] D. Zukowska, A. Melikov, and Z. Popiolek, ‘Impact of personal factors and furniture arrangement on the thermal plume above a sitting occupant’, *Building and Environment*, vol. 49, no. 0, pp. 104–116, 2012.

[28] R. Ramponi and B. Blocken, ‘CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters’, *Building and Environment*, vol. 53, no. 0, pp. 34–48, 2012.

[29] P.-C. Liu, H.-T. Lin, and J.-H. Chou, ‘Evaluation of buoyancy-driven ventilation in atrium buildings using computational fluid dynamics and reduced-scale air model’, *Building and Environment*, vol. 44, no. 9, pp. 1970–1979, 2009.

[30] M. K. Chourasia and T. K. Goswami, ‘CFD simulation of effects of operating parameters and product on heat transfer and moisture loss in the stack of bagged potatoes’, *Journal of Food Engineering*, vol. 80, no. 3, pp. 947–960, 2007.

[31] Z. J. Zhai and Q. Y. Chen, ‘Sensitivity analysis and application guides for integrated building energy and CFD simulation’, *Energy and Buildings*, vol. 38, no. 9, pp. 1060–1068, 2006.

[32] C. Topp, P. Hesselholt, M. R. Trier, and P. V. Nielsen, ‘Influence of geometry of thermal manikins on concentration distribution and personal exposure’, in *The 7th International Conference Healthy Buildings*, Singapore, 2003.

[33] M. Hajdukiewicz, M. Walsh, and M. M. Keane, ‘Formal calibration methodology for a CFD model of a naturally ventilated room’, in *The 12th International Building Performance Simulation Association Conference*, Sydney, Australia, 2011.

[34] J. D. Rambo, ‘Reduced-order modeling of multiscale turbulent convection: Application to data center thermal management’, Georgia Institute of Technology, 2006.

[35] Q. Nie and Y. Joshi, ‘Reduced order modeling and experimental validation of steady turbulent convection in connected domains’, *International Journal of Heat and Mass Transfer*, vol. 51, no. 25–26, pp. 6063–6076, 2008.

[36] Z. Zhai, Z. Zhang, W. Zhang, and Q. Chen, ‘Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 1: Summary of prevalent turbulence models’, *HVAC & R Research*, vol. 13, no. 6, pp. 853–870, 2007.

[37] W. O’Grady and M. Keane, ‘Specification of an IFC based software application to support CFD simulation’, in *The 11th International Conference on Computing in Civil and Building Engineering*, Montreal, Canada, 2006.

[38] P. J. Roache, ‘Perspective: A method for uniform reporting of grid refinement studies’, *Journal of Fluids Engineering*, vol. 116, no. 3, pp. 405–413, 1994.

[39] IRUSE Weather, 2011. [Online]. Available: http://weather.nuigalway.ie/.

[40] Onset Data Loggers, 2011. [Online]. Available: http://www.onsetcomp.com/.

[41] NAP, 2008. [Online]. Available: http://www.tyndall.ie/nap/.

[42] B. O’Flynn, S. Bellis, K. Mahmood, M. Morris, G. Duffy, K. Delaney, and C. O’Mathuna, ‘A 3D miniaturised programmable transceiver’, *Microelectronics International*, vol. 22, no. 2, pp. 8–12, 2005.

[43] Dantec Dynamics, 2011. [Online]. Available: http://www.dantecdynamics.com/.

[44] Ansys 12.1, Ansys Inc., 2009.

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. ‘Formal calibration methodology for CFD models of naturally ventilated indoor environments’. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027

[45] C. Topp, P. V. Nielsen, and D. N. Sørensen, 'Application of computer simulated persons in indoor environmental modeling', *ASHRAE Transactions*, vol. 108, no. 2, pp. 1084–1089, 2002.

[46] I. B. Celik, 'Procedure for estimation and reporting of discretization error in CFD applications', *In Statement on the Control of Numerical Accuracy of the Journal of Fluids Engineering (Editorial Policy), ASME Fluids Engineering Division*, 2004.

[47] ISO 7730:2005, *Ergonomics of the thermal environment - Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria*. International Organization for Standarization, 2005.

[48] ANSI/ASHRAE Standard 55, *Thermal Environmental Conditions for Human Occupancy*. American Society of Heating, Refrigerating and Air-Conditioning Engineers, 2004.

[49] N. Bradley, 'The response surface methodology', Master of Science in Applied Mathematics & Computer Science, Indiana University of South Bend, USA, 2007.

[50] MATLAB v.7.12.0.635 (R2011a), The MathWorks Inc., 2012.

Please reference as:

Hajdukiewicz, M., Geron, M., & Keane, M.M., 2013. 'Formal calibration methodology for CFD models of naturally ventilated indoor environments'. *Building and Environment*, 59, pp. 290-302. http://dx.doi.org/10.1016/j.buildenv.2012.08.027