Numerical And Experimental Study Of A Supply Air Wall Based On Integrated Insulation Clay Hollow Blocks

JB Bouvenot\textsuperscript{1,2} and M Egg\textsuperscript{1}

\textsuperscript{1}National Institute of Applied Sciences of Strasbourg, 24 bd de la Victoire, Strasbourg, France
\textsuperscript{2}ICube Laboratory, 300 bd Sébastien Brant - CS 10413 - F-67412 Illkirch Cedex, France

jean-baptiste.bouvenot@insa-strasbourg.fr

Abstract. Integrated insulation clay hollow blocks is an interesting constructive system in the context of Near Zero Energy Buildings and building energy efficiency. Their simple modification into supply air wall can increase their thermal performance without great effort. This paper deals with the creation of an original supply air wall or window test bench and with the numerical and experimental study of a supply air wall (or ventilated wall) based on modified modern integrated insulation clay hollow blocks where large cavities (about 4 cm) are filled by mineral wool. In some cavities, mineral wool is removed to create a flow pattern, which aims to recover heat losses from inside and solar energy from outside. At first, a 3D CFD numerical model is presented to assess the energy performance of a 1 m\textsuperscript{2} sample of ventilated wall. Then, an experimental test bench, based on a modified guarded hot box simulating solar effects and airflows between the two chambers, is carried out to assess the real performances. A comparison between these two studies allows validating the results which show a good correlation in terms of temperature difference gains between outdoor temperature and pre-heated temperature going up to a maximum of 14 K for only 1 m\textsuperscript{2} of wall and for a volume flow rate of 4 m\textsuperscript{3}/h (4,5 K for 32 m\textsuperscript{3}/h).

1. Introduction

Energy efficiency in buildings is an important lever in the current worldwide energy transition to drastically reduce greenhouse gas emissions and preserve natural resources. In addition, so-called "low tech" systems must be favoured in order to reduce the technical complexity of the systems and to supply: low maintenance requirements, high durability and low life cycle impacts. Supply air walls (windows or opaque walls) are low-tech systems which recover transmission heat losses through the depth of the wall and which catches solar gains to mainly pre-heat new air which is blown indoor. The system described here is a "low tech" system since it is naturally integrated in the structure of the building itself: either via air cavities created by a double or a triple glazing or via cavities structuring blocks as clay hollow blocks or cinder blocks (see Figure 1). Some works already studied double glazed [1] and triple glazed supply air windows [2] [3] [4] [5] or opaque supply air heavy clay bricks walls [6] [7] [8] [9] or opaque supply air clay hollow blocks walls [10] [11] [12] [13]. However, any study dealt with this kind of integrated insulation clay hollow block where the cavities can be freely “activated” on local zones. The idea here is to remove the mineral insulation-filling panel on some cavities to create a ventilated space (flow pattern) within the wall to create an opaque supply air wall (ventilated wall/active wall) using the cavities naturally present that does not require specific building mode (wall construction as usual). The aim being to continuously recover a part of the heat losses through the thickness of the wall on one hand and a maximum of solar gains on the other hand. This pre-heated fresh airflow is then used to renew the air of the building (see Figure 1).
The purpose of this study is to present an experimental and numerical study of an original supply air wall sample concept based on integrated insulation clay hollow blocks at scale 1. This concept aims to do fresh air pre-heating in winter (or pre-cooling in summer) continuously by coupling with a natural or mechanical ventilation system. This system could be as well used as Trombe wall (with inlet and outlet inside to coupled to temperature difference controller to recover solar gains). The study focuses on a sample of 1 m² built between 2 climatic rooms. The numerical study will be carried out via a commercial CFD code (STARCCM+). The experimental study will be carried out thanks to a guarded hot box (2 climatic rooms and a wall sample carrier) modified to generate a solar flux in the “outdoor” chamber and to generate a drop pressure between the “indoor” and “outdoor” chambers to simulate a mechanical ventilation (or a natural ventilation) and to generate a flow within the tested wall. The active wall has been built with CLIMAmur 36® blocks from Wienerberger Company (see Figure 1).

2. Method
The overall method is based on the numerical evaluation of the active wall concept prior to the experimental evaluation. Finally, a validation procedure will be implemented.

2.1. Numerical model

Figure 2 presents the supply wall sample based on air cavities in the wall, air inlets outside at the bottom and air outlet indoor at the top. The number of transfer units (NTU) heat exchanger standard method, as Yu et al. used [1], Seferis et al. [9], Huang et al. [10] or Yu et al. [12] is not suitable here because of the tortuosity of the flow pattern. The aim here is to take into account the impact of the flow pattern on the thermal performances. Other authors implemented more accurate CFD models [6] [11] [15]. The numerical model integrates the real geometry of the test bench. That is why, firstly, the wall sample has been designed in order to ease the construction in the sample carrier by integrating simple air inlets and outlets with connection to the extraction air network. The wall is offset thanks to a cantilever at the bottom (see Figure 2 in purple) from the sample holder to create a cantilever at the bottom in order to allow the air to enter the opening below the wall (see Figure 2). There also is a free
high part only where the 2 rooms are simply delimited by a very thin vertical clay wall. The free space allows recovering the pre-heated air thanks to a convergent plenum (see Figure 2).

Figure 3. Tortuous flow pattern based on 3 cavities (in purple) and CFD velocity magnitude fields.

The wall is built according to the rules of the art that is to say by a staggered stack. Therefore, the shape of the duct for the flow pattern is the consequence of this stack of staggered cavities and generates tortuosity related to singularities of the type shrinkage / widening and bends at 90° (see Figure 3). At the numerical level, the effect of the glue/mortar/joint (obturation and local narrowing) can be taken into account as showed Alghamdi et al. [14] but it will not be taken into account here. It is to notice that only one cavity per row will be activated here. Three cavities were emptied of their mineral insulation (see Figure 3) that means that all the others cavities stay fulfilled by mineral wool. About the main computation parameters about the turbulence model, the fluid properties, the mesh parameters and the solver, at first, the steady state was considered. For turbulence, as Launder [15], Fraisse [16], Gan [17] note, the k-epsilon turbulence model is a suitable model for the flow simulation of circular or rectangular low depth cavities. About the fluid Fraisse et al. [15] assimilate the air to a perfect gas and consider it as an incompressible fluid but obeying the Boussinesq approximation for some others authors [6]. As the flow is considered as forced (forced convection) with a Richardson number below 0,1, a constant air density was considered in the modelling. A rectangular trimmed mesh with a standard wall function has been used [15]. At the air inlet, a uniform speed computed according the volume flow rate set point with a constant temperature (0 °C) has been imposed. At the outlet, atmospheric pressure is assumed. The main computation parameters are summarized in the tables 1 and 2:

### Table 1. Mesh main parameters.

| Area       | Mesh                              |
|------------|-----------------------------------|
| Flow       | Rectangular (Trimmed)             |
|            | Step of 0.004 [m]                 |
|            | Boundary layer (Prism Layer)      |
|            | First step in boundary layer : 0.001 m |
|            | Growing step rate in the boundary layer: 20 % |
|            | Number of cells in the boundary layer : 10 |
| Solid      | Rectangular (Trimmed)             |
|            | Step of 0.0085 [m]                |

### Table 2. Physical model parameters.

| Medium | Physical model |
|--------|----------------|
| Fluid  | Constant density |
|        | Turbulent (model k-epsilon) |
|        | Segregated Fluid Temperature |
|        | Segregated Flow |
| Solid  | Steady state |
|        | Heterogeneous solid |
|        | (clay and mineral wool) |
|        | Segregated Solid Temperature |

Usual thermal properties for each material (fluid and solid) are used as defined in the table 3:
An external temperature $T_{ext}$ corresponding to 0 °C is imposed in order to simulate winter conditions; temperature that the test bench is easily able to reproduce. The internal temperature $T_{int}$ is set to 20 °C. The indoor and outdoor surface heat exchange coefficients (radiation + convection) values $h_{int}$ and $h_{ext}$ are derived from preliminary experimental studies on the guarded hot box (see Table 4). These coefficients are intrinsic to the climatic room. The principle is to simulate on the CFD code the thermal behaviour of the test bench. These values seem to be high due to the mixing of air by fans on both sides but there cannot be changed. Table 4 summarized the boundary conditions and the variation range of experimental parameters:

### Table 4. Boundary conditions.

| Hypothesis                      | Value           |
|---------------------------------|-----------------|
| Outdoor temperature $T_{ext}$   | 273.15 K        |
| Indoor temperature $T_{int}$    | 293.15 K        |
| Indoor surface exchange coefficient $h_{int}$ | 22.5 W/m²/K |
| Outdoor surface exchange coefficient $h_{ext}$ | 22.5 W/m²/K |
| Volume flow range $q_v$         | 4-165 m³/h     |
| Solar flux $\phi_s$             | 0-1000 W/m²     |

#### 2.2. Experimental set up

The original test bench is based on a guarded hot box composed of two 1 m³ climatic chambers and a 1 m² wall sample carrier. By default, the humidity and the temperature can be independently and dynamically controlled. The initial bench has been modified to integrate the possibility to simulate solar effects in the “outdoor” room and to simulate an airflow from outside to inside. Figure 4 shows the test bench with these two modifications.

![Figure 4. Laboratory test bench (on the left) and test bench perspective (on the right).](image)
installation has been modified to integrate a modulating power lamp (0 - 2750 W) simulating a solar insulation in terms of flux density and in terms of spectrum (see Figure 5). The lamp has been selected in order to illuminate the whole wall sample and to produce at least 1000 W/m². A control loop is used thanks to a radiative flux sensor fixed in the middle of the outer face to modulate the power of the lamp according to the set point from 0 to 1000 W/m².

Figure 5. Solar lamp (on the left) and lamp wavelength spectrum.

Then, a ventilation network composed of a fan, an air valve, flexible ducts, a hot wire volume flow meter, and a convergent plenum have been designed and built. The principle is to take air from the laboratory and to inject it in the outdoor chamber by a little opening (cable gland left open). Then the air flows within the wall sample and is extracted by the plenum located in the indoor chamber (see Figure 6) thanks to a modulating speed fan located on the roof of the chambers. A specific device has been designed to pass the duct tightly through the control window in the indoor chamber (see Figure 6). Figures 6 and 7 show respectively the air extraction device at the outlet of the supply air wall (inner face and outer face) and the global air extraction network.

Figure 6. Extraction device at the supply air wall outlet (left) and cantilever creation (on right).

Figure 7. Air extraction network principle scheme.
The wall sample is equipped with different sensors summarized in Table 5:

**Table 5. Metrology features.**

| Metrology                      | Technology                          | Range          | Uncertainty |
|--------------------------------|-------------------------------------|----------------|-------------|
| Temperature                    | K thermocouple                      | -75 - 150 °C   | ± 0.5 K     |
| Global heat flux density       | Double T thermocouple (Captec)      | -∞ – +∞ W/m² | ± 3 %       |
| Radiative flux density         | Double T thermocouple (Captec)      | -∞ – +∞ W/m² | ± 3 %       |
| Volume flow                    | Hot wire (Kimo CTV 120)             | 20-160 m³/h    | ± 3 % ± 1.3 m³/h |

The data logging system is based on a CompactDAQ system from National Instrument. This device is used to control the variable speed fan (0-10 V) and the variable power lamp (2-10 V). The fan speed is controlled according to the flow meter measurement and its set point. The lamp control signal is adjusted according to a radiative flux-meter which measures the incoming flux density on the outer face of the wall. All the measurements and the emitted control signals are managed in Labview® digital environment.

### 3. Result

At first, the repeatability of the experimental results has been investigated to confirm the reliability of the measurements. Two tests have been carried out three times and the temperature increase between the wall inlet (outside air: point A on Figure 2) and the wall outlet (pre-heated air: point B on Figure 2) has been measured. The repeatability results are shown in Table 6:

**Table 6. Repeatability tests.**

| Solar flux density [W/m²] | Air volume flow [m³/h] | Temperature difference between inlet and outlet of the wall [K] | Max. deviation [K] |
|--------------------------|------------------------|---------------------------------------------------------------|-------------------|
| 1000                     | 32                     | 3.18                                                          | 0.35              |
|                          |                        | 3.53                                                          |                   |
|                          |                        | 3.45                                                          |                   |
| 400                      | 32                     | 1.72                                                          | 0.82              |
|                          |                        | 2.55                                                          |                   |
|                          |                        | 2.07                                                          |                   |

**Figure 8.** Temperature gains according to air flow rate.

**Figure 9.** Temperature gains according to solar flux density.

The maximum deviation for these two repeatability tests is below or equivalent to the uncertainty range. The results are considered as acceptable and all the future tests will be done once. Then an
experimental campaign has been launched by studying mainly the impact of solar radiation and air volume flow on this supply air wall. The main monitored quantity is the pre-heated air temperature at the outlet of the system. Besides, experimentally, it is difficult to reach very low flows below 5 m$^3$/h because of the flow meter technologies and ranges mainly. Therefore, the volume flows range goes from 30 to 130 m$^3$/h (the larger range possible). To know the wall behaviour at lower volume flows, the CFD model was used as soon as it will be validated. For the solar flux density, the classical range: 0-1000 W/m² has been used. Figure 8 shows the result in terms of temperature gains between the outdoor air and the pre-heated air at the outlet of the ventilated wall considering a solar flux density of 1000 W/m². The trend of each curve is similar with CFD results, which are within the uncertainty range of measurements. There is a significant difference between the results from the CFD model and the experimental results, but this difference remains within the limits of the absolute composed uncertainty on the measured temperature difference (< 0.7 K). As expected, the gains are higher for low airflows that is logical because the presence time in the ventilated wall (which behaves as a heat exchanger) is higher, that allows the air to reach higher temperatures. At maximum, the temperature gain is about 4 K that is equivalent to a recovery sensible heat gain of 48 W per activated cavity. Figure 9 shows the result in terms of temperature gains considering a volume flow of 32 m$^3$/h. The results between experimental and CFD outputs are very close except for low solar flux densities. The maximum temperature gain is about 3.5 K for the maximum solar flux density. To explain the gap at low solar flux density, it can be noticed that the new airflow induced by the additional fan interferes with the internal flow behavior of the outdoor chamber. By punctual measurement, lower temperatures were measured at the inlet of the ventilated wall in comparison with the mean temperature of the chamber in contact with the wall. It appears like stratification or a dead zone (recirculation zone) phenomenon that maintains a colder air layer at the bottom of the chamber. In comparison with the CFD model, which considers a same reference temperature for the outside air and the wall inlet air temperature (0 °C), in the test bench, there is a temperature gap between the air at the bottom of the cold box (at the inlet of the ventilated wall) and the mean value of the air in the box (gradients/stratification) that leads to artificially increase the temperature gain. It was hard to obtain a more homogeneous temperature repartition in the climatic chamber. Besides, there is an uncertainty on the solar flux absorption coefficient of the wall which has been set to 0.9 in the CFD model. Finally, in practice, low air flows induce high temperature gains were observed that’s why it would be interesting to investigate the thermal performances for lower air flows. Instead of using only 1 cavity, several cavities could be ventilated in parallel to increase the global ventilation flow by maintaining very low flows in each cavity. A CFD simulation computed a temperature gain of 14 K considering a solar flux of 1000 W/m² and a volume flow rate of 4 m$^3$/h/cavity. This configuration reaches recovery rates of 19 W/cavity (4.8 W/(m$^3$/h)) on only 0.75 m of height (3 rows of bricks). The number of activated cavities would depend on fresh air requirement according to each room (activity, number of people).

4. Conclusion
A specific test bench has been created to experimentally study supply air wall or windows. New devices have been implemented to simulate solar effects and wind or mechanical effects by respectively a modulating solar lamp and an air extraction network equipped by a variable speed fan. Then, an innovative “low tech” supply air wall has been designed and studied. This ventilated wall is based on integrated insulation hollow clay blocks which one of the cavities has been removed from its mineral insulation to allow airflow. At first, a numerical CFD model has been carried out emulating the 1 m² wall prototype built in a guarded hot box. Then, an experimental campaign has been led to assess the thermal performances. The results show a good correlation between the numerical model and the experimental device with temperature gains thanks to the recovery of heat losses and solar flux from 1 to 4.5 K in the volume flow range [30-130 m$^3$/h]. For low air flows (up to 4 m$^3$/h), the CFD model shows temperature gain going until 14 K. Several perspectives are considered. The first step will be to study a scale change by considering at least a stage (about 3 m high) or an entire facade. Then, a parametric study on the position (outer/middle/inner cavity?) of the activated cavities would be interesting. Experimentally, the blocks were not sealed. A future work has to be led to assess the impact of joint/mortar, which seals each block together, and ensure the structure strength and lift. This
additional medium can strongly modify or obturate the flow pattern [14]. Then, yearly building energy simulations using TRNSYS for example would be interesting to assess the energy relevance of this ventilated wall. Several cases could be investigated: heat recovery mechanical ventilation coupling or natural ventilation coupling. Finally, the strength of this concept is its simple and integrated design (“low tech”) which thermally increases a system without complicating it by materials or system adding. Therefore, a life cycle analysis on this wall concept would be relevant to compare it with standard building concepts.

Acknowledgments
We thank the Wienerberger Company for their technical and logistic support, the Arconic Company for their financial support and Professor Siroux for giving us access to the guarded hot box.

References
[1] Bhamjee M, Nurick A and Madyira DM 2013 An experimentally validated mathematical and CFD model of a supply air window: Forced and natural flow E. & B. 57 289-301
[2] Gloriant F, Tittelein P, Joulin A and Lassue S 2015 Modeling a triple-glazed supply-air window B. & Env. 84 1-9
[3] Gloriant F, Tittelein P, Joulin A and Lassue S 2015 Study of the Performances of A Supply-Air Window for Air Renewal Pre-Heating Energy Procedia 78 525-30
[4] Baker PH and McEvoy M 2000 Test cell analysis of the use of a supply air window as a passive solar component Solar Energy 69 Issue 2 113-30
[5] Michaux G, Greffet R, Salagnac P and Ridoret JB 2019 Modelling of an airflow window and numerical investigation of its thermal performances by comparison to conventional double and triple-glazed windows Applied Energy 242 27-45
[6] Buratti C, Palladino D and Di Palma R 2018 Development and optimization of a new ventilated brick wall: CFD analysis and experimental validation E. & B. 168 284-97
[7] Fantucci S, Serra V and Perino M 2015 Dynamic Insulation Systems: Experimental Analysis on a Parietodynamic Wall Energy Procedia 78 549-54
[8] Dimoudi A, Androustopoulos A and Lykoudis S 2004 Experimental work on a linked, dynamic and ventilated, wall component E. & B. 36 Issue 5 443-53
[9] Seferis P, Strachan P, Dimoudi A and Androustopoulos A 2011 Investigation of the performance of a ventilated wall E. & B. 43 Issue 9 2167-78
[10] Huang J, Yu J and Yang H 2018 Effects of key factors on the heat insulation performance of a hollow block ventilated wall Applied Energy 232 409-23
[11] Yu J, Yang J and Xiong C 2015 Study of dynamic thermal performance of hollow block ventilated wall Renewable Energy 84 145-51
[12] Yu J, Huang J, Xu X, Ye H, Xiong C, Wang J and Tian L 2017 A semi-dynamic heat transfer model of hollow block ventilated wall for thermal performance prediction E. & B. 134 285-94
[13] Fantucci S, Marinosci C, Serra V and Carbonaro C 2017 Thermal Performance Assessment of an Opaque Ventilated Façade in the Summer Period: Calibration of a Simulation Model through in-field Measurements Energy Procedia 111 619-28
[14] Alghamdi AA and Alharthi HA 2017 Multiscale 3D finite-element modelling of the thermal conductivity of clay brick walls Construction and Building Materials 157 2017 1-9
[15] Launder BE 1984 Turbulence models and their applications 2 Eyrolles
[16] Fraisse G, Boichot R, Kouyoumji JL and Souyri B 2010 Night cooling with a Ventilated Internal Double Wall E. & B. 42 Issue 3 393-400
[17] Gan G 2011 General expressions for the calculation of air flow and heat transfer rates in tall ventilation cavities, Building and Environment 46 Issue 10 2069-80
[18] Bouvenot JB, Jimenez V, Desport L and Siroux M, 2019, Numerical And Experimental Thermal Inertia Characterization Of An Integrated Insulation Clay Hollow Block For Buildings Thermal Comfort Applications Roma IBPSA Conference September 2019