Scientific Paper / Artículo Científico

UNIVERSIDAD POLITÉCNICA SALESIANA ECUADOR
INGENIUS
Revista de Ciencia y Tecnología
https://doi.org/10.17163/ings.n20.2018.04
pISSN: 1390-650X / eISSN: 1390-860X

Prediction of thermal impact reduction in a double wall building

Predicción de la reducción del impacto térmico en un edificio con doble pared

Marcelo Eduardo Berli[1,*], Agustín Brondino[1], José Di Paolo[1]

Abstract

In Santa Fe de la Vera Cruz city, Argentina, a building that includes elements of sustainable architecture, energy efficiency and comfort based on the use of natural resources is being built. Specifically, a double facade design on the front walls is meant to achieve an air chamber that prevents heat transfer from the outside to the inside in summer and vice versa in winter. In this work, a numerical study is presented for the evaluation of the thermal performance of a cavity (air chamber) interposed in a double facade of the building for different climatic conditions, considering two air chambers alternatives: connected and non connected to the outside. Both cases are energetically compared with the standard facade design without chamber. The results show that for summer conditions, a chamber connected to the outside would be the most efficient design, while for winter, the closed cavity is the best saving-energy alternative.

Keywords: computer simulation, energy saving, environment conditioning, sustainable architecture.

Resumen

En la ciudad de Santa Fe de la Vera Cruz, Argentina, se está construyendo un edificio de altura que incluye elementos de arquitectura sustentable, eficiencia energética y confort logrado con la utilización de recursos naturales. Particularmente, un diseño de doble fachada en los frentes que dan al exterior para lograr una cámara de aire que impida la transferencia térmica desde el exterior al interior en verano y al revés en invierno. Este trabajo presenta un estudio numérico de la evaluación del desempeño térmico de la cavidad interpuesta en la doble fachada del edificio, para distintas condiciones climáticas, considerando dos alternativas de diseño: cámara de aire cerrada y cámara de aire conectada con el exterior. Ambos casos se comparan con la situación de inexistencia de la cámara, cuya transferencia de energía térmica se constituye en el caso patrón. Los resultados muestran que para las condiciones de verano, la cavidad con conexión al exterior sería la más recomendable, mientras que para el invierno, la cavidad cerrada es más apta para el ahorro de energía.

Palabras clave: acondicionamiento de ambientes, arquitectura sustentable, ahorro de energía, simulación computacional.

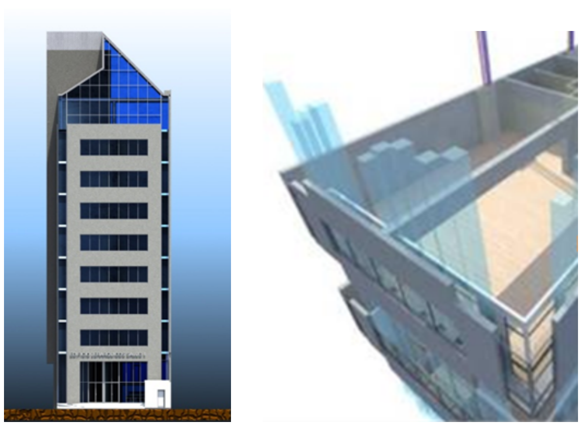
[1,*]PID-UTN AMUTIFE 3457, Research Group in Fluid Mechanics, Universidad Tecnológica Nacional, Santa Fe, Argentina. Autor para correspondencia ✉: mberli@santafe-conicet.gob.ar,
https://orcid.org/xxxx-xxxx-xxxx-xxxx, https://orcid.org/xxxx-xxxx-xxxx-xxxx
https://orcid.org/xxxx-xxxx-xxxx-xxxx

Received: 14-05-2018, accepted after review: 18-06-2018

Suggested citation: Berli, M. E.; Brondino, A. and Di Paolo, J. (2018). «Prediction of thermal impact reduction in a double wall building». Ingenius. N.°20, (july-december). pp. 39-47. doi: https://doi.org/10.17163/ings.n20.2018.04.

1. Introduction

The Jerárquicos Salud mutual society of the city of Santa Fe de la Vera Cruz, Argentina, is building a high-rise administrative building that includes elements of sustainable architecture, is energy efficient and achieves comfort based on the use of the greatest amount of natural resources possible (Figure 1-a). For this purpose, the east, west and southern sides were designed with a double façade that runs from the ground floor to the top floor, which consists of an external and an internal wall of different materials, both separated by a 50 cm thick space of air, as seen in the detail of Figure 1-b. In this way, an air chamber is created that physically separates the external and internal facade and has the objective of achieving thermal insulation between the exterior and the interior of the building. The air chamber of each floor communicates with the chambers of the upper and lower floors by means of circulation holes made in the slab.



(a) (b)

Figure 1. a) Scheme of the building with double facade. b) View in section and perspective of the double facade.

Because the air chambers of each floor are interconnected, there is a possibility that an air flow can be generated that totally or partially covers the height of the building. This flow would be beneficial from the thermal point of view, especially for the summer periods, since air circulation acts as a barrier that reduces heat transmission from the external side to the internal and transports a significant amount of thermal energy towards the outside of the building, avoiding its entry. The translation in thermal comfort and reduction of the energy consumption in the interior is direct, regulating, in addition, the environmental conditions of the workspaces with greater effectiveness.

However, the potential thermal and energy benefits of the current design, its effective operation and possible modifications required as the work progresses in order to to optimize thermal behavior, are not directly predictable, requiring at this stage experimental and predictive and/or computational tools.

An experimental study of the interconnected air chambers and their thermal performance requires a high investment in materials, time and human resources. On the other hand, the studies carried out by computational simulation yield numerical predictions whose results which help obtain inferences that orient experimentation towards more accurate values. Numerical results guide the design, and the success of their predictive power is based not only on them being based in physical laws, but also on their ability to adapt to new ideas and explore a large number of alternatives.

This work presents the numerical study of the thermal performance of a cavity interposed in the double facade of the building, through computer simulations for different climatic conditions, considering two design alternatives: closed air chamber and air chamber with connection to outside air currents.

The results show that for summer conditions, the design of the cavity implies a significant reduction in thermal energy that would enter the building. Among the alternatives analyzed, the designs of the cavity with connection to the outside would be the most recommended in summer and closed chamber would be the most suitable for winter.

2. Materials and methods

The work is constituted as a computational theoretical work, based on hypotheses about the phenomenon of thermal transfer in the air chamber produced by the double façade. These considerations are summarized in turbulent flow and thermal transfer dominated by convection [1–4] and are listed below:

1) Stationary state because the atmospheric conditions to which the building is exposed vary very slowly during a day, this approach is acceptable and used for the most demanding conditions of the summer and winter seasons.

2) The thermal transfer between the floors occurs only through the circulation holes, assuming that the slabs are perfect insulators. This means that, when calculating the thermal energy that enters each floor, said energy can only come from sources that are connected to the chamber, that is, from the outside and from the air chambers of adjacent floors.

3) There is no contribution of thermal energy by artifacts, people, lights or other sources. This simplification is done to study only the energy savings that result from the existence of the air chamber.

4) The contribution of thermal energy by radiation from the external wall to the internal one is neglected.

5) The moisture content of the air circulating in the cavities is negligible.

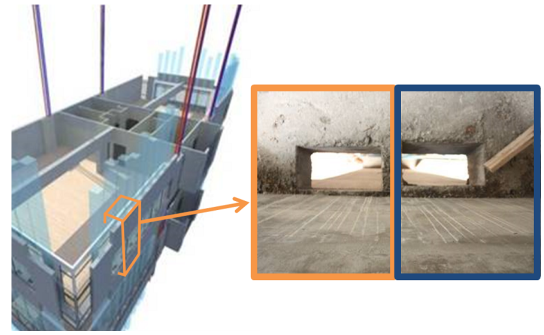
6) The air flow in the cavities develops in a turbulent regime. Because the goal is not to have detailed information of the boundary layer in the contact between the walls and the air, a turbulence model of the k-ε type was used, applied in an advanced simulation software.

7) The internal and external walls are assumed to be smooth.

2.1. Definition of the area where the simulations will be carried out

As mentioned in the introduction, this work consists in the study of a physical model that is representative of the cavity whose thermal performance is to be studied. As is known, the availability of computer tools with high computing capacity facilitates the solution of complex problems such as the one addressed in this work. However, since the availability of resources is limited, the size of the problem under study must be reduced in such a way that it is solvable and that, at the same time, ensures the portion studied is representative of the whole problem. In the case of the cavity under study, the simulation of the problem in all of its dimensions is computationally very expensive. For this reason, it is possible to section the problem in a portion whose dimensions contain all the geometric characteristics that condition the air flow in the chamber, so that the behavior of the rest of the cavity can be considered as a repetition of the portion studied. The selection of said portion can be seen in Figure 2-a (transparent prism of orange edges). The prism that delimits the selected area consists of a portion of the chamber corresponding to any floor of the building. If a a photograph of a top view of said chamber was taken, the result would be an image like the one shown in the orange box of Figure 2-b, where the presence of the circulation hole made in the corresponding floor slab and connecting the air chamber with the one of the previous floor can be observed.

As observed in Figure 2-b, the adjacent rectangular section (blue box) and the selected one (orange box) are arranged so as to mirror each other. From a physical point of view, this fact implies a symmetry in the geometry, indicating that the solution of the problem in the orange box section is the same as in its contiguous section (blue box), but mirrored. Mathematically speaking, this means that the derivatives of the variables involved, with respect to the horizontal direction, are null. Thus, the scheme of the two holes in Figure 2-b is repeated with the same positions, each slab dividing the two adjacent floors.



(a) (b)

Figure 2. a) Portion selected for the simulation. b) Circulation holes made in the slab.

As there is a repeated scheme, it is acceptable to solve the problem in the selected portion with standard computational resources. If the interior and exterior wall portions are added to this selection, between which the air chamber and the holes in the slabs are located, the definition of a simulation module is reached, which can be seen in Figure 3. The dimensions and materials of each module are the following:

- External wall: built of concrete, 20 cm thick, 3.45 m high and 70 cm wide.
- Internal wall: built of concrete, 7 cm thick and other dimensions equal to the external wall.
- Air chamber: 50 cm thick and other dimensions equal to the outer wall.
- Holes: 20 cm × 40 cm horizontal section and a thickness (in the slab) of 25 cm.

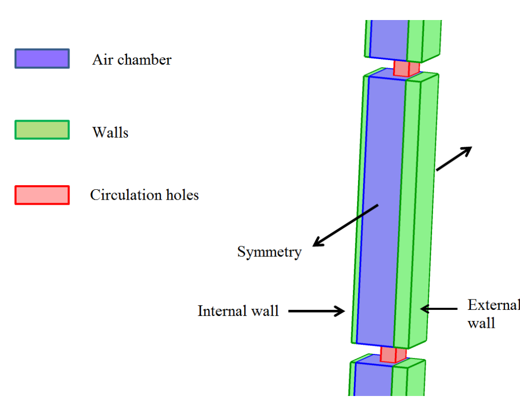


Figure 3. Geometric diagram of the simulation module.

The sum of all the modules through the circulation holes will define the total geometry for the simulation of the problem to be solved. That is, 8 modules like the one in Figure 3, interconnected by the circulation holes.

2.2. Model equations and resolution methodology

The movement of air inside the cavity is mainly due to flotation forces associated with the density gradients that are caused by the difference in temperatures between the walls that generate the cavity. This phenomenon is known as natural convection and its dynamics have been described in previous works by means of the Boussinesq approximation. This approximation considers density variations only in volumetric forces, by means of a linear function with the change in temperature and with validity for incompressible laminar flow and low thermal gradients.

In the simulated cavity, thermal gradients are usually higher than the limits of validity of the Bousinesq approximation [5]. For this reason, and in order to move towards more realistic simulations, a compressible flow model is used in this work, assuming turbulent flow in a stationary state and neglecting the effects of radiation on energy transfer. In this way, the differential equations that describe the flow of natural convection are as follows:

Continuity

$$\nabla \cdot (\rho \mathbf{u}) = 0 \qquad (1)$$

Amount of movement

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot\Big[-P\mathbf{I} + (\mu+\mu_T)(\nabla\mathbf{u}+(\nabla\mathbf{u})^T) - \frac{2}{3}(\mu+\mu_T)(\nabla\cdot\mathbf{u})\mathbf{I} - \frac{2}{3}\rho k\mathbf{I}\Big] + (\rho-\rho_0)\mathbf{g} \qquad (2)$$

Energy

$$\rho C_p \mathbf{u}\cdot\nabla T = \nabla\cdot(k_A\nabla T) \qquad (3)$$

where $\mathbf{u}$ is the velocity vector, ρ is air density, $\rho 0$ is air density at ambient temperature (external), P is the modified pressure, g is the acceleration of gravity, μ is air viscosity, μ_T is turbulent viscosity, k is turbulent kinetic energy, C_p is the air's heat capacity, k_A the thermal conductivity of the air and T the temperature.

To describe the turbulent flow, the k-ε model was used, which has been shown to be the most accurate for the calculation of air movement inside rooms in houses and buildings [6]. However, it should be noted that its precision decreases very close to the walls, where models such as the k-ε of low Reynolds number promise a better description of the velocity and temperature profiles [7]. However, this study aims to show the general benefits of the system and not an exact prediction of the values that are calculated near the walls, in which case the description of the k-ε model is very useful for a first estimation and leads to results that require much less computational cost and are more quickly obtained, the latter being desirable in studies required for the decision making of construction companies. Thus, in addition to the conservation equations, the following are added: the equation (4) of turbulent kinetic energy variation and the equation (5) of turbulent dissipation velocity:

$$\rho(\mathbf{u}\cdot\nabla)k = \nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_k}\right)\nabla k\right] + P_k - \rho\epsilon \qquad (4)$$

$$\rho(\mathbf{u}\cdot\nabla)k = \nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_\epsilon}\right)\nabla\epsilon\right] + C_{\epsilon 1}\frac{\epsilon}{k}\Big(\mu_T\cdot\Big[\nabla\mathbf{u}:(\nabla\mathbf{u}+(\nabla\mathbf{u})^T) - \frac{2}{3}(\nabla\cdot\mathbf{u}^2)\Big] - \frac{2}{3}\rho k\nabla\cdot\mathbf{u}\Big) - C_{\epsilon 2}\rho\frac{\epsilon^2}{k} \qquad (5)$$

Turbulent viscosity μ_T is defined by Equation (6).

$$\mu_T = C_\mu \rho \frac{k^2}{\epsilon} \qquad (6)$$

The parameters of equations (4) to (6) are considered constant, with the following values [5]:

$$C_{\epsilon 1} = 1,44$$
$$C_{\epsilon 2} = 1,92$$
$$C_\mu = 0,09$$
$$\sigma_k = 1$$
$$\sigma_\epsilon = 1,3$$

2.2.1. Condiciones de contorno

Contour conditions

The k-ϵ turbulent flow model used does not solve the profile of velocities against the solid wall but uses wall functions that model the high velocity and temperature gradients that occur in that zone. For this, in addition to the water impermeability condition defined by Equation (7) and the null kinetic energy flow, defined by Equation (8), Equation (9) using a wall function was implemented:

$$\mathbf{u}\cdot\mathbf{n} = 0 \qquad (7)$$

$$\nabla k\cdot\mathbf{n} = 0 \qquad (8)$$

$$\Big[(\mu+\mu_T)(\nabla\mathbf{u}+(\nabla\mathbf{u})^T) - \frac{2}{3}(\mu+\mu_T)(\nabla\cdot\mathbf{u})\mathbf{I} - \frac{2}{3}\rho k\mathbf{I}\Big]\mathbf{n} = -\rho\frac{u_\tau}{\delta_w^+}\mathbf{u}_{tang} \qquad (9)$$

With

$$\epsilon = \rho \frac{C_\mu k^2}{K_v \delta_w^+ \mu}$$

Where $\mathbf{u}_{tang}$ is the tangential velocity, defined as $\mathbf{u}_{tang} = \mathbf{u} - (\mathbf{u} \cdot \mathbf{n})\mathbf{n}$, u_τ is friction speed, K_v is the Von Kármán constant and δ_w^+ is the dimensionless thickness of the wall function. In addition, normal voltage equal to zero was assumed in the inlet and outlet holes, $\tau \cdot \mathbf{u}$, and the pressure at the lower entrance was arbitrarily defined as equal to 0.

Constants in each simulation, the temperatures of the walls that delimit the cavity, and the conditions of symmetry in the cuts that are seen in Figure 3 were considered for the thermal analysis. The temperature of the air entering the cavity is also assumed to be constant and equal to the ambient temperature (T_a).

The differential equations presented, along with their corresponding boundary conditions, are solved numerically using the finite element method with the COMSOL 4.4 commercially licensed software.

2.2.2. Verification of the flow regime

Previous works [1–4, 8, 9] have addressed the analysis of the phenomenon of natural convection in representative air cavities (ceilings, among others) by laminar flow models. However, due to the dimensions of the cavities in this work and the properties of the air, it is reasonable to expect that under normal conditions the flow may be turbulent, which had to be corroborated prior to the selection of the model. For this purpose, the Reynolds number and the Grashof number were used as dimensionless parameters, which were estimated using the physical properties of dry air presented in Table 1. These properties were adopted for normal atmospheric pressure (10^5 Pa) and 30 °C [10], corresponding to the ambient temperature (reference) used in the model.

Table 1. Physical properties of dry air at 30°C and atmospheric pressure

Description	Value
Density (ρ)	1,205 [kg/m^3]
Viscosity (μ)	1,82e-5 [N s/m^3]
Thermal conductivity (k_A)	0,0257 [W/(m K)]
Heat capacity (C_p)	1,005 [kJ/(kg K)]
Coefficient of thermal expansion (β)	3,43e^{-3} [1/K]

The Reynolds number compares the relationship between inertial forces and viscous forces and is defined by the ratio $R_e = \rho U_c L/\mu$, withU_c and L representing the speed and length characteristic to the model. On the other hand, the Grashof number indicates the relationship between the flotation forces and viscous forces, and is defined by the relation $Gr = \rho^2 g \beta \Delta T L^3/\mu^2$, where ΔT is the temperature difference characteristic of the system under study. When the flotation forces are, by comparison, higher than the viscous forces, the regime is considered to be turbulent. The transition between these two regimes for vertical plates is given for a Gr of the order of 10^9 [10].

If the floating flow velocity $U_c = (g\beta \Delta T L)^{1/2}$, the length $L = 3m$ equal to the height of each chamber between two contiguous floors, and the temperature difference between the two walls of the double facade, $\Delta T = 56$ $°C$ are defined as characteristic parameters, the resulting values of $Re \approx 10^6$ and $Gr \approx 10^{11}$ clearly indicate the existence of a turbulent or transition regime, but not laminar. Thus, the results presented in the following section correspond to an air flow inside the cavity that is in a turbulent regime for all the simulated conditions.

3. Results and discussion

The first case demanding study is that of extreme conditions in summer. The section of the external wall in contact with the outside has a temperature of 70 °C, assuming it is exposed to the incidence of the sun in the hours of maximum temperature. On the other hand, the section of the inner wall in contact with the interior of the building has a temperature considered to be pleasant for a working environment, that is, 24 °C. As the objective is to study the thermal performance of the cavity, it is reasonable to first consider the situation of absence of cavity, assuming the external and internal walls are in direct contact, and also suppressing the circulation holes. It should be noted that, in the case of an absence of cavity, both walls are conserved, since having only the external wall would mean that the results would be modified not only by the inclusion of the cavity but also by the addition of another wall, in which case the analysis could not focus only on the existence or absence of the cavity.

3.1. Witness situation: absence of cavity

Figure 4 shows a diagram of the conditions in which the simulation was performed.

Because there are no circulation holes, consideration 2) of section 2 allows for the calculation of the thermal energy transfer of one module independent of the others. Given this situation, a thermal energy input of 204 W/m^2 has been calculated in each module, a value which will be used in the following cases. It should be clarified that, unlike the coming cases, the only thermal transmission mechanism is conduction. For this reason the obtained value can be checked using the Fourier law.

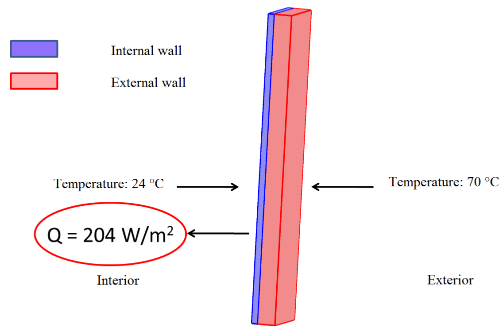


Figure 4. Simulation conditions and thermal energy for absence of cavity.

3.2. Cavity without connection to the exterior

The second case analyzed consists of the air chamber without connection to the outside. That is, although the chambers of all the floors are interconnected through the holes, none of the chambers has any opening that connects them to the external environment of the building. The air will thus be trapped in the chambers and there can only be circulation through the holes. The temperature difference between the external and internal walls generates variations in air density and therefore natural convection [8,9] as shown in Figure 5.

It can be observed that the air in contact with the hottest wall (in red) rises and increases its temperature (the green areas imply a higher temperature than the light blue ones), while in the vicinity of the cold wall (in blue) the air descends and the temperature decreases. In this way, there is a movement of air by the physical phenomenon of natural convection, by which the air transmits thermal energy not only by conduction, but also by convection.

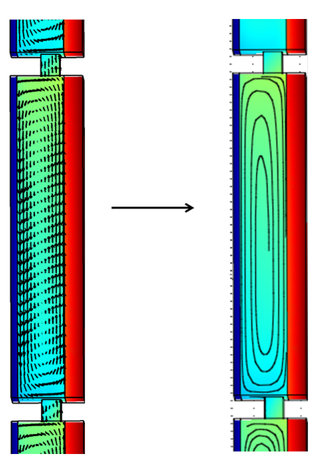


Figure 5. Side view of a module showing air circulation. Left: velocity vectors. Right: current lines.

When quantifying the heat transmitted for this scheme, the resulting value is 51 W/m^2, which means that the reduction of thermal energy that would enter from the outside is 75% with respect to the values of section 3.1.

Given this result, it can be questioned whether the reduction obtained is reasonable. In order to answer this question, it can be assumed that the air trapped in the chamber was stagnant without performing recirculation movements, in which case the energy transferred to the interior would be 2.3 W/m^2 (calculation that can also be performed with Fourier's law), implying a reduction of 99%. These values are in accordance with the fact that air is a poor heat conductor, having a thermal transmission coefficient of 0.025 (W/mK) in comparison, for example, with concrete whose coefficient is 1.5 (W/mK), 60 times higher than that of air. The fact that the reduction is 75% and not 99% is because the movement of the air transfers additional heat by convection.

It should be noted that since there is no connection with the outside, it was found that the air recirculates inside a module and, therefore, there is no flow through the circulation holes, there is no transfer of energy between the modules. This results in the thermal energy flow to the interior being identical in each module. Of course, this is an idealized situation in which all floors are at the same internal temperature and consequently there are no differences in temperatures that could cause a convective movement between the floors.

This result indicates that the mere existence of the air chamber can result in significant energy savings when maintaining a pleasant internal environment.

3.3. Cavity with connection to the outside and natural flow

To simulate the connection of the chambers with the exterior, holes of similar dimensions are assumed to be the circulation holes in the outer walls of the first and last floor modules. That is, section holes of 20 cm by 40 cm and the thickness of the outer wall (20 cm). For the first floor, the hole was designed in the lower lateral part of the first module, as shown in Figure 6-a, while for module 8 the hole was made in the upper lateral part, as shown in 6-b.

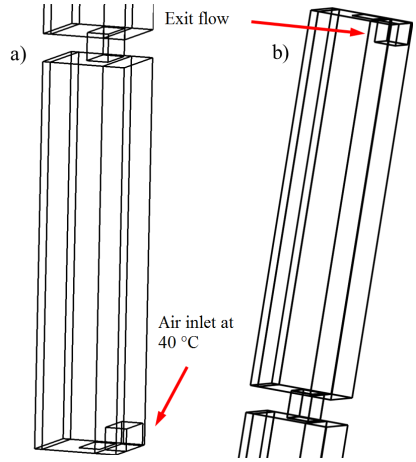


Figure 6. a) Hole that connects the lower module with the exterior. b) Hole that connects the upper module with the exterior.

To carry out the simulation, the same conditions as for case 3.2 were considered, but since in this case there may be air entry from the outside, it is assumed that it is at an elevated summer temperature of 40 °C. The conditions of entry and exit in the first and last module respectively, are indicated in Figure 6, while for the remaining modules the external and internal temperatures are maintained.

The presence of connections with the outside enable a flow that covers all the floors, from the first to the last. Indeed, the results indicate the existence of this flow, as seen in Figure 7. This figure shows, in a side view of the side furthest from the circulation orifice, that the air maintains some recirculation leading to similar effects as in case 3.2, but the side view near the orifice shows the existence of a flow that passes through the cavity from the lower module to the upper one. This circulation would be a way of venting the cavity to avoid the possible stagnation of humidity and the generation of bad odors.

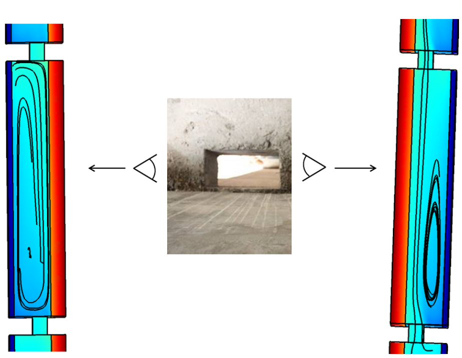


Figure 7. Circulation diagrams by natural convection in the cavities.

As for energy saving, the existence of circulation between the modules modifies the individual performance of each one. The lower modules are benefited since the flow of air from the outside absorbs a certain amount of thermal energy from the hottest wall and transports it by convection to an upper module, so that the higher the modules, the higher amount of energy they receive from the air absorbed from the lower modules.

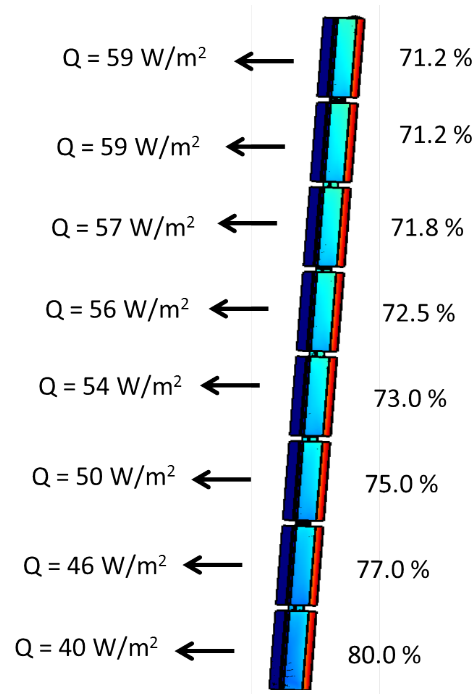


Figure 8. Thermal performance of the 8 modules. The percentages indicate the reduction in the heat transfer of each floor with respect to the situation without an air chamber.

Figure 8 shows the thermal energy transfers and the reduction percentages for each module with respect to case 3.1. It can be observed that the first three modules have a thermal performance equal to or higher than in the case 3.2, however, the savings decrease. Overall, the average reduction between the 8 modules is 74%, very similar to the previous case. In addition, these results can be useful at the time of plotting the occupation of each floor.

4. Simulations for winter conditions

The presence of the cavity, according to the numerical results of this study, shows a very good thermal performance in summer conditions, but it remains to be seen if this behavior is representative of conditions in a winter day. For this purpose, the temperatures of the external wall and of the air were modified, which, in the case of the cavity with connection to the outside, would enter from outside. It should be mentioned that for winter conditions, energy flows outwards, since the situation reverses with respect to summer. In addition, the geometry pattern that arises from subtracting the cavity to have a comparison reference is maintained. The boundary conditions are as follows:

Internal wall temperature: 24 °C.
External wall temperature: 10 °C.
Ambient air temperature: 10 °C.

This condition would imply a poor incidence of the sun on the external wall and therefore it remains at the same temperature as the outside air.

For the cavity with connection to the outside and natural flow, the results outlined in Figure 9 show that the reduction in energy losses from the interior to the atmosphere is 63%. Although it is an acceptable value, the circulation of air at a lower temperature in contact with the inner wall implies an absorption of thermal energy from the inside. Before this physical fact, the case of closed cavity was analyzed, and the results are schematized in Figure 10. The savings obtained for this second case are 85%, significantly higher than the previous one. It is therefore advisable that, in winter conditions, there is no circulation between the cavities and the exterior.

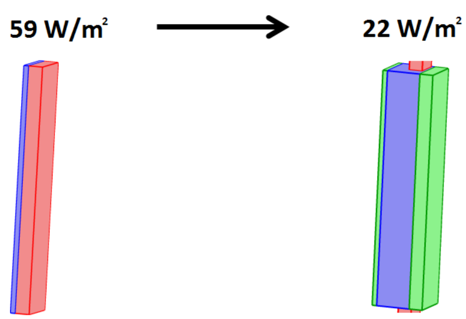


Figure 9. Comparison between absence of cavity and cavity with natural ventilation.

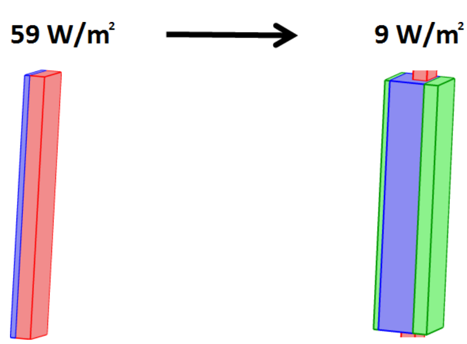


Figure 10. Comparison between absence of cavity and closed cavity.

5. Conclusion

This research presents the numerical study of the thermal performance of a cavity interposed in the double façade designed for the building of the Jerárquicos Salud mutual society in the city of Santa Fe de la Vera Cruz, Argentina. The analysis was made based on computer simulations of different climatic conditions and two design alternatives were considered according to the scheme provided: closed cavity and with connection to the exterior.

The results show that, for summer conditions, the design of the cavity implies a significant reduction in the thermal energy that would enter the building. Both the closed cavity and the cavity connected to the outside are valid alternatives that showed thermal aptitudes for the reduction of transferred energy, with an estimated reduction in summer of around 75% with respect to a design without the cavity. However, the case of the cavity connected to the outside, due to natural ventilation, would be the one selected because it has the possibility of renewing the air trapped in the cavities, reducing the possibility of accumulating humidity and bad odors.

Regardless of the promising predictions for the cavity with natural ventilation for the summer, the results have shown that its thermal performance in winter is inferior with respect to the closed cavity scenario. In the case of natural ventilation in winter, a reduction in thermal energy lost to the exterior of 63% was estimated, while for the design of a closed cavity, this reduction would be of 85%.

The final conclusion of this study based on computational predictions, is that it is suggested to make the orifices of circulation with a system of air flow control, which allows the option to keep the orifices open during the days of higher temperatures and closed during the periods of lower temperatures. Future works with more accurate predictive models near the walls will allow for more precise adjustments in the predictions of this study in terms of the calculated values, while these results can be used conceptually since it is estimated that the accuracy of the heat transfer calculations will not change the trends shown in this work.

Finally, it is worth noting that, according to all the numerical studies carried out, the mere presence of the air chamber shows a remarkable improvement of the thermal performance in terms of energy transfers between the atmosphere and the interior of the building, with the suggestion of the previous paragraph showing the best performance.

References

[1] J. Hirunlabh, S. Wachirapuwadon, N. Pratinthong, and J. Khedari, "New configurations of a roof solar collector maximizing natural ventilation," *Building and Environment*, vol. 36, no. 3, pp. 383–391, 2001. DOI: https://doi.org/10.1016/S0360-1323(00)00016-0.

[2] M. Ciampi, F. Leccese, and G. Tuoni, "Energy analysis of ventilated and microventilated roofs," *Solar Energy*, vol. 79, no. 2, pp. 183–192, 2005. DOI: https://doi.org/10.1016/j.solener.2004.08.014.

[3] H. Asan and L. Namli, "Laminar natural convection in a pitched roof of triangular cross-section: summer day boundary conditions," *Energy and Buildings*, vol. 33, no. 1, pp. 69–73, 2000. DOI: https://doi.org/10.1016/S0378-7788(00)00066-9.

[4] H. F. Oztop, Y. Varol, and A. Koca, "Laminar natural convection heat transfer in a shed roof with or without eave for summer season," *Applied Thermal Engineering*, vol. 27, no. 13, pp. 2252 –2265, 2007. DOI: https://doi.org/10.1016/j.applthermaleng.2007.01.018.

[5] A. Brondino, M. E. Berli, and J. Di Paolo, "Aislación térmica producida por cavidades de aire. análisis de un techo con geometría simplificada y flujo turbulento," in *V Congreso Argentino de Ingeniería Química, Santiago del Estero, Argentina.*, 2016. [Online]. Available: https://goo.gl/2bVmLv

[6] C. Ghiaus, F. Allard, M. Santamouris, C. Georgakis, and F. Nicol, "Urban environment influence on natural ventilation potential," *Building and Environment*, vol. 41, no. 4, pp. 395–406, 2006. DOI: https://doi.org/10.1016/j.buildenv.2005.02.003.

[7] S. Wang, Z. Shen, and L. Gu, "Numerical simulation of buoyancy-driven turbulent ventilation in attic space under winter conditions," *Energy and Buildings*, vol. 47, pp. 360–368, 2012. DOI: https://doi.org/10.1016/j.enbuild.2011.12.012.

[8] M. E. Berli, J. Di Paolo, and F. A. Saita, "Mecánica de fluidos computacional aplicada al diseño térmico pasivo de techos," in *I Congreso Argentino de Ingeniería Mecánica, Bahía Blanca, Argentina*, 2008.

[9] ——, "Heat transfer on a naturally cross-driven ventilated triangular cavity with openings," *Journal of Physics: Conference Series*, vol. 166, no. 1, pp. 1–10, 2009. DOI: http://doi.org/10.1088/1742-6596/166/1/012019. [Online]. Available: http://stacks.iop.org/1742-6596/166/i=1/a=012019

[10] A. Bejan, *Convection Heat Transfer, Fourth Edition*, I. John Wiley & Sons, Ed., 2013. DOI: http://doi.org/10.1002/9781118671627.