Recibido 31/08/2021
Aceptado 09/12/2021

CFD SIMULATION OF AN INCLINED ROOF SOLAR CHIMNEY

SIMULACIÓN MEDIANTE CDF DE UNA CHIMENEA SOLAR INCLINADA SOBRE LA CUBIERTA

SANDRA RODRÍGUEZ-TREJO
Magíster en Diseño Bioclimático
Estudiante doctoral, Posgrado en Diseño Bioclimático,
Dpto. de Medio Ambiente
Universidad Autónoma Metropolitana Azcapotzalco
Ciudad de México, México
https://orcid.org/0000-0002-6139-4985
sandraelisar@gmail.com

VÍCTOR FUENTES-FREIXANET
Doctorado en Diseño, Arquitectura Bioclimática
Académico Investigador,
Universidad Autónoma Metropolitana Azcapotzalco
Ciudad de México, México
https://orcid.org/0000-0002-7426-2391
ffva@azc.uam.mx

RESUMEN
En los últimos años se ha elevado el consumo energético derivado del uso de aparatos eléctricos para promover el movimiento de aire en regiones con climas cálidos, lo que ha traído como consecuencia un impacto negativo en el ambiente. En este trabajo se evalúa el desempeño de una chimenea solar utilizada para inducir la ventilación natural en un espacio cerrado, bajo las condiciones del clima cálido mexicano. Para tal efecto, se desarrollaron simulaciones CFD empleando el modelo de turbulencia RNG k- e y el modelo de radiación DO, y considerando únicamente los fenómenos de convección natural. El desempeño de la chimenea solar se evaluó comparando los resultados de las simulaciones con mediciones experimentales; análisis que reveló una buena concordancia. Se obtuvieron temperaturas de hasta 46.5°C en el aire dentro de la chimenea y de 77.1°C en la placa de absorción; resultados que permiten verificar la influencia del fenómeno de descarga de calor por flotación natural del aire en la chimenea.

Palabras clave
energía solar, ventilación, simulación CFD

ABSTRACT
In recent years, energy consumption from electrical devices to foster air movement in regions with warm climates has risen, with the resulting negative impact on the environment. The purpose of this paper is to evaluate the performance of a solar chimney used to induce natural ventilation in a closed space, under the weather conditions of the hot humid Mexican climate. For this purpose, CFD simulations were run using the RNG k- e turbulence model and the DO radiation model, considering only natural convection phenomena. The solar chimney performance was evaluated, comparing the results of the simulations with experimental measurements, analysis which showed a good match. Temperatures of up to 46.5%°C in the air within the chimney, and of 77.1°C on the absorption plate, were obtained, results that allow verifying the influence of the heat discharge phenomenon by the natural flotation of air in the chimney.

Keywords
solar energy, ventilation, CFD simulation

INTRODUCTION

Around 30% of residential electricity consumption in Mexico is used to attain thermal comfort in warm climate areas (González & Beele, 2016), a proportion that continues to noticeably increase as the use of air-conditioning equipment rises. The amount of electricity used for artificial climatization in warm climates depends on several factors, specifically the design and elements of the space's envelope, as well as the equipment that provides the three elements that determine thermal comfort: temperature, humidity, and airspeed (de Buen, 2017).

The results of the National Survey on Energy Consumption in Private Housing (National Institute of Statistics and Geography [INEGI, in Spanish], 2018), show there are 14.6 million fans in the country. The ones most commonly used in dwellings, known as ceiling and pedestal fans, total 11.5 million, with a preponderant use, where for 34.4% of the dwellings, they are used between 5 and 9 hours a day.

These data provide a general overview of energy consumption from devices that promote air movement in warm climates, and their resulting economic and environmental impact. From this comes the importance of implementing passive climatization strategies to favor natural ventilation and to provide thermal comfort.

Natural ventilation in buildings comes from two sources: by wind pressure (cross ventilation), and by temperature differences, namely, from the air density between the outside and inside (Stack effect). In warm humid climates, ventilation through the Stack effect is inefficient, since the temperature delta between the outside and inside of naturally ventilated buildings, is limited. In this context, solar chimneys represent an efficient alternative to promote natural ventilation, thanks to their ability to take advantage of solar resources to heat one part of the building, fostering the extraction of warm air by convection.

A solar chimney is basically a solar air heater that is embedded, vertically or horizontally, on a building as part of a wall or roof, and whose classification can vary depending on the different setups or functions (Lal, Kaushik & Bhargav, 2013). Different variations can be found in solar chimney designs. These are affected by several factors, such as the location, climate, orientation, size of the space being ventilated, and internal heat gains. However, the basic elements involved, like the solar collector, the transparent roof, and the entry and exit openings, are part of each design (Khanal & Lei, 2011).

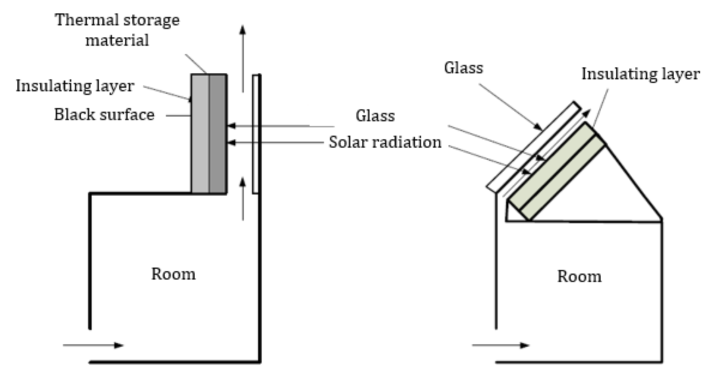

Figure 1. Solar chimney setups. Source: Harris & Helwig (2007, p. 136).

CHARACTERISTICS AND OPERATION OF SOLAR CHIMNEYS ON THE ROOF

To promote natural ventilation, solar chimneys inclined on the roof or cover work with the same principles as vertical chimneys, that is to say, they are assisted by the suction effect on openings located on the lower part of the chimney, and the hot air that is extracted through the upper openings of the flue (Figure 1). Generally, solar chimneys are made up of a transparent cover, and an absorption or thermal storage plate, separated by a cavity or air flue. They have an insulating layer on the back of the plate, to avoid heat losses.

In most cases, the transparent cover of the solar chimney is made of glass, a material that is transparent to short-wave radiation. One part of the incident solar radiation on the surface of the glass is transmitted directly to the internal environment, another is reflected outside, and a third is absorbed by the material. As the glass has low absorption and is exposed to heat exchanges by convection on its two faces, its surface temperature is always close to that of the air it is in contact with.

The absorption plate is commonly made of a metallic material with high emissivity and absorbance values and is heated by absorption from the portion of solar radiation transmitted by the glass. A part of this heat may be lost by conduction towards the external side of the plate, where there is thermal insulation, and another part is ceded to the air in the cavity by thermal convection, and to the glass, by infrared radiation. In this way, with the increase in temperature of the air present in the chimney's cavity, which is heated by the absorption plate and the glass, a rising air movement is generated towards the upper opening.

The strategy of integrating solar chimneys throughout the slope of the roof or cover of the buildings allows having large areas to collect solar irradiance and taking advantage of it when the sun is at its highest. Some disadvantages of these systems are that the effective height is restricted by the angle of inclination of the roof and the possibility of greater pressure losses due to changes of direction, apart from the air's path (Harris & Helwig, 2007).

PREVIOUS RESEARCH PROJECTS

A significant amount of research has been made since the 1990s, through experimental, analytical, and computational methods, focusing especially on finding optimal design solutions to improve natural ventilation using solar chimneys. According to Shi *et al.*, 2018), the performance of these systems is mainly based on eight factors: the angle of inclination; the cavity width; the height; the dimensional ratio between height/cavity; the entry opening area; the exit opening area; the entry/exit ratio, and solar radiation. Zhang, Tan, Yang, and Zhang (2016) also mention the thermal characteristics of the solar radiation absorption material.

The angle of inclination of solar chimneys is one of the most analyzed parameters. Several research projects report that an inclination of 45° is optimal to obtain a maximum airflow (Chen *et al.*, 2003; J. Mathur, Bansal, S. Mathur, Jain & Anupma, 2006; Bassiouny & Korah, 2009; Saifi, Settou, Dokkar, Negrou & Chennouf, 2012; Jianliu & Weihua, 2013; Al-Kayiem, Sreejaya & Gilani, 2014; Saleem, Bady, Ookawara & Abdel-Rahm, 2016; Dhahri & Aouinet, 2020; Kong, Niu & Lei, 2020), an inclination that generates few pressure losses, as it represents a balance between the Stack pressure and the heat transfer by convection (Yusoff, Salleh, Adam, Sapian & Sulaiman, 2010; Mahdavinejad, Fakhari & Alipoor, 2013).

The recommendations for the sizing of the cavity thickness vary between 0.10m and 0.35m. Several studies agree that airflow is reduced with the increase in cavity thickness, due to the presence of inverse flow. In research made by Shi *et al*. (2016), Saifi *et al*. (2012), Yusoff *et al*. (2010) y Zhai, Dai & Wang (2005), no inverse flow is recorded in cavities of 0.20 m. Meanwhile, Bouchair (1994), Li, Jones, Zhao, and Wang (2004), and Liping and Angui (2004) report a 1:10 ratio between the cavity width and the effective height of the chimney, as optimal.

The works done on the increase in length of the solar chimney have shown a lower impact of this geometric characteristic on chimney performance, compared to the increase in the cavity width. The range of chimney lengths analyzed in previous research runs from 0.40 m to 3.00 m. In this sense, Neves (2012) states that after 1.50 m, the variations in length result in progressively lower increases in the airflow.

In general, the range of the setup variables analyzed in previous studies is limited. These focus mainly on cavity width and inclination angle. The optimal values suggested do not apply to all solar chimney setups, as they have reciprocal dependence relationships between two or more variables, so it is necessary to carry out more research to evaluate their performance under different contexts.

DYNAMIC OF COMPUTATIONAL FLUIDS

Computational Fluids Dynamics (CFD), is a branch of fluid mechanics that uses data structures and numerical methods to solve and analyze the equations that govern the flow of fluids, while also allowing quantitatively and qualitatively knowing their behavior.

CFD simulation is one of the most important methods to study natural ventilation and can be used to predict airflow and heat transfer inside and outside buildings. CFD models are capable of providing a detailed spatial distribution of the air temperature, pressure, speed, and turbulence (Sánchez, 2017).

One of the most used numerical techniques to discretize the equations that describe the movement of fluids (Navier-Stokes) is the method of finite volumes, whose starting point is the breakdown of the dominion to be analyzed in a finite number of adjoining control volumes, where the variables are calculated in the nodes located in the centroid of each control volume. The control volumes and the nodes are defined using a numerical mesh, which is essentially a discrete representation of the dominion where the problem is solved.

The description of the CFD simulation model used in this research, to analyze airflow through a solar chimney, is presented in the following section. It is worth mentioning that the boundary conditions were determined based on experimental results obtained from the onsite monitoring of a life-size physical model.

METHODOLOGY

The work presented here has a quantitative approach (using mathematical, statistical, and IT tools). Likewise, the decision was made to run an experimental study and a numerical analysis to collect and analyze the data, seeking causal relationships between the measured variables. The analytical scientific method has been followed regarding the analysis of the simulation results, and the comparison of numerical and experimental results.

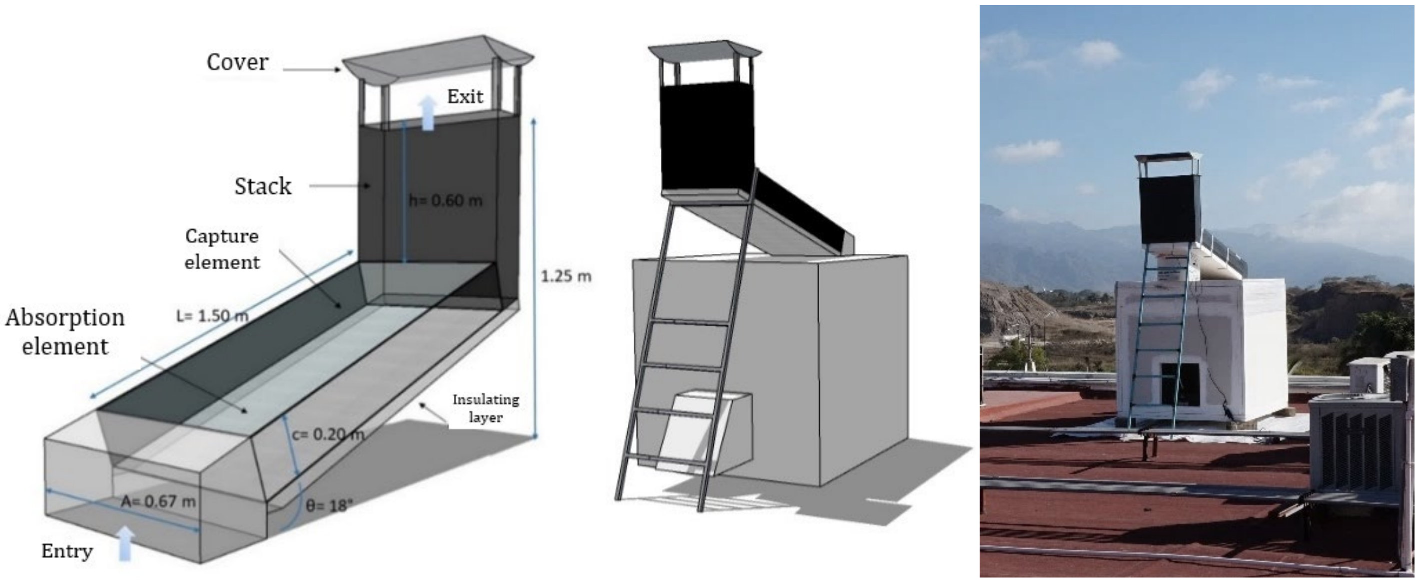

Figure 2. Characteristics of the experimental model. Source: Preparation by the authors.

The design of the research starts from two fundamental phases. The first of these is the design of the solar chimney using the analysis of the existing bibliography, the determination of experimentation periods, and the construction characteristics of the experimental prototype. The second phase consisted in applying the CFD technique to simulate the system, to later validate the CFD model used from the experimental results, and to check whether a coherent solution has been found for the problem.

EXPERIMENTAL STUDY

The experimental study was carried out under the climate conditions of the city of Puerto Vallarta, Mexico, with coordinates 20°C 42' North and 105° 13' West, at 10.0 masl. The climatic variables, including the room temperature, relative humidity, wind direction and speed, and global irradiance, were acquired from an automatic meteorological station.

The geometric setup of the solar chimney being analyzed was designed, establishing an inclination of 18° towards the South to obtain suitable solar irradiance conditions throughout the year, at the latitude of the study. A black opaque 22 caliber galvanized steel sheet collector was built, of 1.50 m in length, 0.67 in width, and 0.20 m of cavity thickness. To offset the pressure loss caused by the low inclination angle, a vertical extension or stack of 0.60 m was placed, coupled to the collector.

The space being ventilated consisted of a cubic chamber made of plaster panels of 1.50 per side, where a square opening of 0.50 m per side was placed, on the lower part of the vertical face opposite the collector entry, at a distance of 0.15 m from the base. A simple 4 mm thick clear glass cover was used as a solar radiation capturing element, and a flat 2.0 mm thick black opaque aluminum sheet as an absorption element, insulated on its inside face with a 5 cm thick polystyrene plate. The characteristics of the experimental model are presented in Figure 2.

Sensors were placed on the entrance of the chamber and inside it, inside the collector, on the absorption plate's surface, and at the exit of the stack. The variables to monitor were the following:

- Airspeed in the chamber entrance.
- Temperature inside the chamber, plate temperature.
- Temperature of the air inside the cavity.
- Airspeed at the collector entrance.
- Speed at the stack exit.

Data were recorded every 5 minutes for three days in different months throughout the year, and hourly averages were then calculated, and statistical analyses were made to determine the data to be used in the simulations.

CFD SIMULATION

The methodology used to run simulations in most CFD codes contains three main stages: pre-processing; solution; and post-processing. Likewise, each one of these stages consisted of the following activities:

Revista Hábitat Sustentable Vol. 11, N°. 2. ISSN 0719 - 0700 / Págs. 72 - 81
https://doi.org/10.22320/07190700.2021.11.02.06

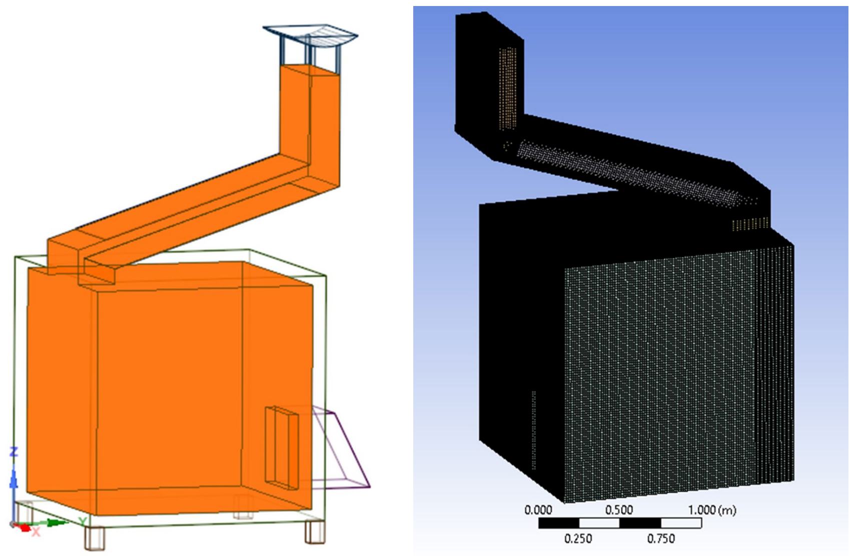

Figure 3. 3D model and analysis mesh. Source: Preparation by the authors.

Pre-processing:

- Choice of the suitable calculation model to solve the specific problem and to determine the target of the simulation.
- Definition of the geometry the problem represents.
- Generation of a suitable mesh, considering its dimensions and topology, as well as the independence of the results of the mesh used.
- Establishing the properties of the fluids.

Processing or solution:

- Determination of the boundary conditions, such as the temperature of the surfaces, the heat flow, the fluid entries and exits, the heat sources and sumps, moisture, etc.
- Definition of the solution's algorithms.
- Selection of the suitable turbulence model.
- Accuracy of the numerical parameters like differentiation schemes, relaxation factors, number of iterations, convergence criteria, etc.

Post-processing:

- Obtaining data, graphs, vectors, flow lines, outlines, etc. of the variables of interest.
- Validation of the model, comparing results with experimental data and other numerical results.

The commercial software ANSYS Fluent was used to develop the model and the numerical analysis. A 3D model was chosen (Figure 3) to make the solar chimney simulations, with the flow in a stationary state, considering only the natural convection phenomena. To simulate the airflow throughout the model, the parameters were adjusted to obtain the same conditions of the experimental test, considering a turbulent, incompressible, and stationary airflow.

Given the setup of the solar chimney's experimental physical model, a structured mesh was chosen, formed by hexahedra, whose sizes were setup considering the dimensions of the chimney, making sure that its shape complied with the requirements, so that the numerical calculations were stable and precise: high orthogonal quality, and low obliqueness of the elements. The mesh was refined in the areas of interest, like the glass cover, the aluminum plate, and the air entry and exit. With the meshing process, seven meshes were obtained, with between 660,711 and 2,376,021 elements, which were configured and tested to determine the suitable number of elements for the model, allowing reducing computational costs without losing information quality. A mesh formed by 1,633,428 elements was chosen for its analysis.

The Energy equation was activated, to allow calculating the parameters related to heat transfer and density variety. The Viscosity model was also activated with the RNG k–e turbulence model, which is described as stable, given that it provides reasonably accurate results for most of the indoor airflows (Rordigues, Frick, Bejat & Garrecht, 2021). The Discrete Ordinates (DO) radiation model, which allows simulating radiation through semitransparent mediums, was chosen, as well as radiation issues with non-participating mediums (like air), where the radiation indirectly affects the flow field, changing the layout conditions on the surfaces. Likewise, the Semi-Implicit Method of Pressure Linked Equations (SIMPLE) was chosen, as the coupling algorithm of pressure and speed.

Finally, the residual value of 10^{-6} was set as a convergence criterion for the energy equation, and of 10^{-3} for the remaining variables, as suggested in

Revista Hábitat Sustentable Vol. 11, N°. 2. ISSN 0719 - 0700 / Págs. 72 - 81
https://doi.org/10.22320/07190700.2021.11.02.06

Material	Density ρ	Specific Heat Cp	Thermal conductivity λ	Absorbance	Emissivity
	kg/m3	J/kg·K	W/m·K	α	E
Glass	2500	750	1.40	0.14	0.85
Aluminum	2719	871	202.40	0.97	0.98

Table 1. Thermal-physical and optical properties of the materials. Source: Preparation by the authors.

the ANSYS user manual.

The system operates at sea-level atmospheric pressure, so the preset value of 101,325 Pa was kept. The operating temperature is the room temperature recorded during different experimentation periods. For all the cases being simulated, it was considered that the air enters the chamber perpendicularly at the entry opening, at a constant speed of 0.15 m/s, and the temperature values obtained during the experimental test were introduced. Table 1 shows the physical properties of the materials of the elements that make up the model, and that take part in the energy exchange.

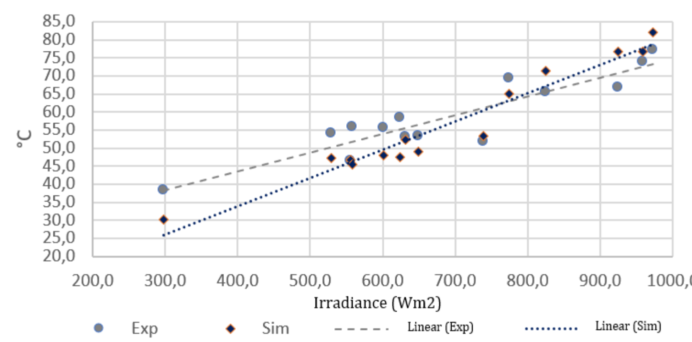

Figure 4. Aluminum plate temperature. Preparation by the authors.

RESULTS AND DISCUSSION

The Pearson linear correlation coefficient was used to analyze the results, to establish similarities between simulated variables and experimental results. Figure 4 shows the spread of the plate temperature data obtained both in experimental monitoring and the simulations. The linear correlation coefficient between the data series equals 0.93, which reveals a high positive correlation. The correlation between irradiance and plate temperature obtained in the simulations is r=0.97, while in the experimental results, this is r=0.91.

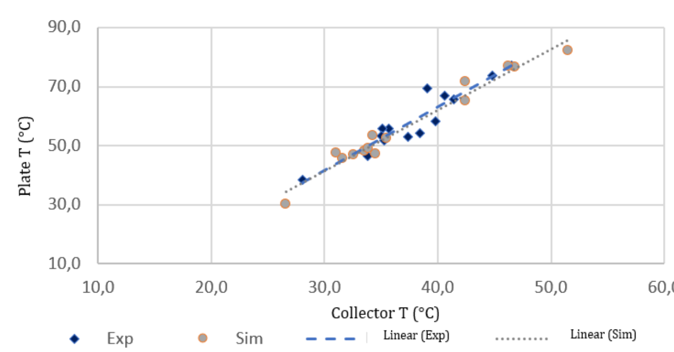

Figure 5. Correlation between air temperature and plate temperature.

The air temperature inside the chimney collector is directly related to the plate temperature, due to the heat transfer by convection (Figure 5); the correlation coefficient between these variables in the experimental measurements is r=0.94, and in the simulations, r=0.98, which indicates a high positive correlation.

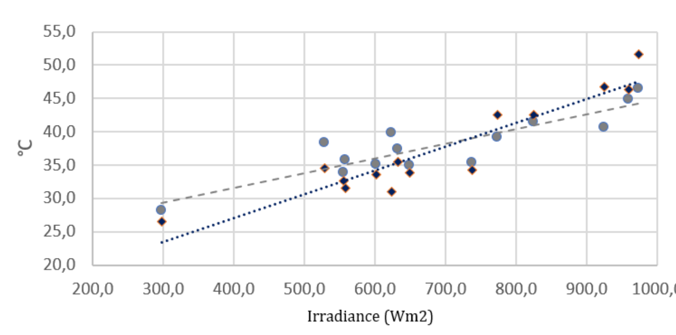

Figure 6. Correlation between air temperature and irradiance. Source: Preparation by the authors.

In an analysis of the spread of air temperature data inside the collector, it is seen that the temperature obtained in the simulations is higher than the experimental data when irradiance conditions are higher than 750 W/m² (Figure 6). The correlation coefficient between both collector temperature data series (experimental and simulations) is r=0.87.

The results of the comparison of speeds at the slack exit between the experimental and simulation

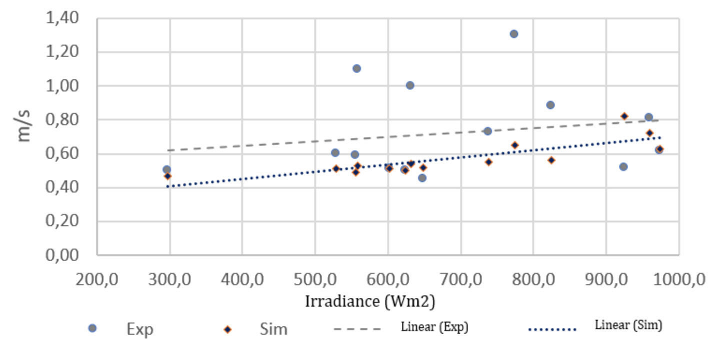

Figure 7. Correlation between air temperature and irradiance. Source: Preparation by the authors.

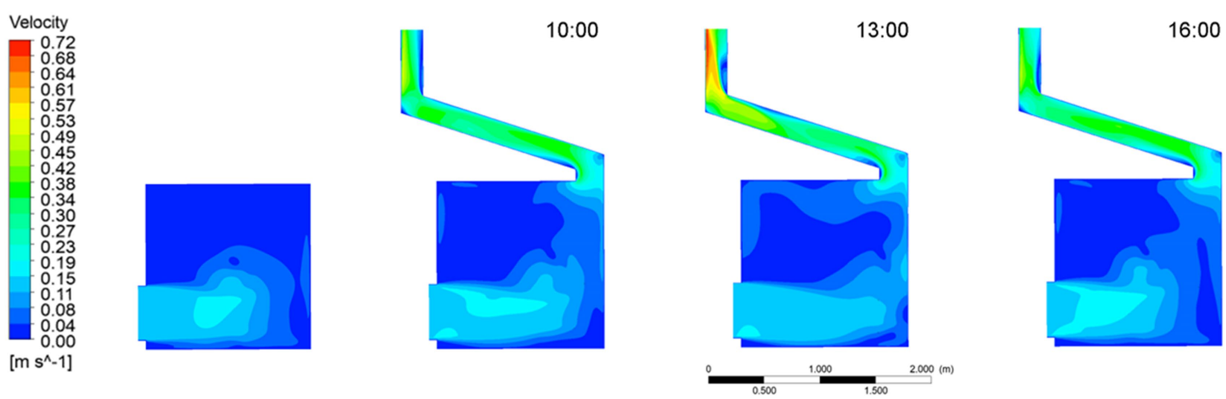

Figure 8. Airspeed layouts in the model's medium plane. Source: Preparation by the authors.

measurements, under different irradiance conditions, are presented in Figure 7, where the difference between the results is noticeable. The correlation coefficient between the experimental measurements and the simulation results is r=0.23.

This is explained because the CFD model does not consider the influence of the wind which, according to that stated in the experimental measurements, has a positive effect on the airspeed at the slack exit.

The internal airspeed is another variable considered when evaluating natural ventilation. The increase of air movement has the potential to change the thermal acceptability at higher operation temperature values (Saadatjoo, Mahdavinejad & Zhang, 2018). To analyze this variable, speed layouts were obtained on the model's medium plane. Figure 8 illustrates the layouts generated by the simulation made with experimental data in October, which had the highest amount of solar irradiance available within the warmest period in the study area. This corresponds to a monthly irradiance average of 363 W/m^2 and a mean temperature of 27.6°C. The changes in the airspeed distribution can be seen in three moments of the day. After comparing the information obtained with the control simulation (the chamber without chimney and exit opening), higher air circulation is seen at 1 pm, close to the solar midday.

CONCLUSION

The results of this research show that it is feasible to use a vertical extension in inclined chimneys to offset the pressure loss caused by the reduction of the inclination angle, and thus optimize solar gains. A smaller inclination would facilitate the incorporation of the system to covers with slopes no greater than 20%, as is the case of most constructions for residential use in the study area.

The data obtained in the simulations showed a good fit with the experimental measurements: the CFD model was capable of suitably predicting the absorption plate temperatures, with a correlation coefficient of r=0.93

with the experimental results. The temperatures inside the collector also showed a high positive correlation with the experimental measurements (r=0.87).

With the CFD simulations, it is possible to verify the importance that the heat discharge phenomenon by natural flotation of hot air has in this system, both for the experimental results and for those obtained by simulation. In fact, a strong correlation of the irradiance with the airflow of the chimney was seen. The CFD analysis allowed studying the airflow patterns within the chimney, by showing the speed layouts.

After comparing the CFD results with the experimental results, the conclusion has been reached that the computational model suitably described the thermo-fluid dynamic behavior of the solar chimney, using the turbulent model k-e RNG complemented with the Discrete Ordinates radiation model.

The seasons and times of the day where natural ventilation is most needed as a thermal comfort strategy in the study area, coincide with the favorable conditions to provide high airflow rates, which states the advantage of using this chimney setup for natural ventilation in low-latitude regions with warm humid climates. It is considered that, by using solar chimneys, it is possible to improve ventilation without economic and energy expenses, as the system does not need conventional energy and does not generate operation or maintenance expenses.

The predictions for speed, temperature, humidity, contaminant concentration, flow patterns, and air exchange rates in buildings are required to design healthy and comfortable indoor environments. The approach proposed in this research contributes to the practical use of the CFD method in architectural design, providing detailed information about the numerical model used. The satisfactory results of these simulations allow concluding that it is possible to simulate and study 3D cases in a stationary state affordably and simply.

Natural ventilation is not only crucial for energy conservation and the reduction of carbon emissions but also to improve the comfort level and the air quality of the built environment. Recently, the importance of providing sufficient ventilation in closed spaces has been pointed out, to promote indoor air renewal with outdoor air, and to reduce the transmission of sicknesses.

From the analyses made, it is concluded that the setup of the proposed solar chimney has significant potential when it comes to increasing natural ventilation in places where cross ventilation cannot be used, bearing in mind that generally rooms of a dwelling are characterized on being ventilated on a single face. It is recommended, for later studies, to make 3D simulations at a life-size scale, and to analyze the temperature distribution in a room of a dwelling with the inclusion of doors, windows, thermal loads, and so on.

ACKNOWLEDGMENTS

This work was carried out within the thesis program "*Caracterización de un colector solar de aire con tiro térmico en cubierta para la mejora de la ventilación natural en clima cálido subhúmedo*" (Characterization of a Solar Thermal Air Collector to Improve Natural Ventilation in a Sub Humid Warm Climate, in English), of the PhD Program on Bioclimatic Design, of the Arts and Science Division for Design of the Autonomous Metropolitan University. Thanks are given to the Centro de Estudios Meteorológicos de la Costa (CEMCO, in Spanish).

BIBLIOGRAPHICAL REFERENCES

Al-Kayiem, H., Sreejaya, K.V. y Gilani, S. (2014). Mathematical analysis of the influence of the chimney height and collector area on the performance of a roof top solar chimney. *Energy and Buildings*, *68*, 305-311. DOI: http://dx.doi.org/10.1016/j.enbuild.2013.09.021

Bassiouny, R. y Korah, N. (2009). Effect of solar chimney inclination angle on space flow pattern and ventilation rate. *Energy and Buildings, 41*, 190-196. DOI: 10.1016/j.enbuild.2008.08.009

Bouchair, A. (1994). Solar Chimney for Promoting Cooling Ventilation in Southern Algeria. *Building Services Engineering Research and Technology*, *15*, 81-93.

Chen, Z., Bandopadhayay, P., Halldorsson, J., Byrjalsen, C., Heiselberg, P. y Li, Y. (2003). An experimental investigation of a solar chimney model with uniform wall heat flux. *Building and Environment, 38*, 893-906.

Dhahri, M. y Aouinet, H. (2020). CFD investigation of temperature distribution, air flow pattern and thermal comfort in natural ventilation of building using solar chimney. *World Journal of Engineering*, *17*, 78-86. DOI: https://doi.org/10.1108/WJE-09-2019-0261

De Buen, O. (2017). *El confort en la demanda eléctrica*. Recuperado de https://www.energiaadebate.com/blog/2338/

González, L. y Beele, A. (2016). *Estudio de caracterización del uso de aire acondicionado en vivienda de interés social*. Comisión Nacional para el Uso Eficiente de la Energía. Recuperado de https://www.gob.mx/cms/uploads/attachment/file/164221/Conuee_2016_Estudio_Caracterizaci_n_Uso_Aire_Acondicionado_en_Vivienda.pdf

Harris, D. y Helwig, N. (2007). Solar chimney and building ventilation. *Applied Energy, 84*, 135-146. DOI: 10.1016/j.apenergy.2006.07.001

Instituto Nacional de Estadística y Geografía [INEGI] (2018). *Encuesta Nacional de Consumo de Energéticos en Viviendas Particulares (ENCEVI) 2018*. Recuperado de https://www.inegi.org.mx/contenidos/programas/encevi/2018/doc/encevi2018_presentacion_resultados.pdf

Jianliu, X. y Weihua, L. (2013). Study on solar chimney used for room natural ventilation in Nanjing. *Energy and Buildings, 66*, 467-469. DOI: http://dx.doi.org/10.1016/j.enbuild.2013.07.036

Khanal, R. y Lei, C. (2011). Numerical investigation of the ventilation performance of a solar chimney. *ANZIAM Journal, 52*, 899-913. DOI: https://doi.org/10.1016/j.enbuild.2011.03.035

Kong, J., Niu, J. y Lei, C. (2020). A CFD based approach for determining the optimum inclination angle of a roof-top solar chimney for building ventilation. *Solar Energy, 198*, 555-569. DOI: https://doi.org/10.1016/j.solener.2020.01.017

Lal, S., Kaushik, S. y Bhargav, P. (2013). Solar chimney: A sustainable approach for ventilation and building space conditioning. *International Journal of Development and Sustainability, 2*(1), 277-297. Recuperado de: https://isdsnet.com/ijds-v2n1-20.pdf

Li, A., Jones, P., Zhao, P. y Wang, L. (2004) Heat transfer and natural ventilation airflow rates from single-sided heated solar chimney for buildings. *Journal of Asian Architecture and Building Engineering 3*(2): 233-238. DOI: 10.3130/jaabe.3.233

Liping, W. y Angui, L. (2004). A numerical study of vertical solar chimney for enhancing stack ventilation in buildings. En *Proceedings of the 21th conference on passive and low energy architecture, Eindhoven, The Netherlands* (pp. 1–5). Recuperado de http://alexandria.tue.nl/openaccess/635611/p1145final.pdf

Mahdavinejad, M., Fakhari, M. y Alipoor, F. (2013). The Study on Optimum Tilt Angle in Solar Chimney as a Mechanical Eco Concept. *Frontiers of Engineering Mechanics Research, 2*(3), 71-80. Recuperado de https://www.researchgate.net/publication/256198688_The_Study_on_Optimum_Tilt_Angle_in_Solar_Chimney_as_a_Mechanical_Eco_Concept

Mathur, J., Bansal, N., Mathur, S., Jain, M. y Anupma. (2006). Experimental investigations on solar chimney for room ventilation. *Solar Energy, 80*, 927-935. DOI: 10.1016/j.solener.2005.08.008

Neves, L. (2012). *Chaminé solar como elemento indutor de ventilação natural em edificações*. Tesis doctoral. Universidade Estadual de Campinas. Recuperado de http://repositorio.unicamp.br/jspui/handle/REPOSIP/258774

Rodrigues, N., Frick, J., Bejat, T. y Garrecht, H. (2021). Using CFD to Evaluate Natural Ventilation through a 3D Parametric Modeling Approach. *Energies, 14*(8). DOI: https:// doi.org/10.3390/en14082197

Saadatjoo, P., Mahdavinejad, M. y Zhang, G. (2018). A study on terraced apartments and their natural ventilation performance in hot and humid regions. *Building Simulation, 11*, 359–372. DOI: https://doi.org/10.1007/s12273-017-0407-7

Saifi, N., Settou, N., Dokkar, B., Negrou, B. y Chennouf, N. (2012). Experimental study and simulation of airflow in solar chimneys. *Energy Procedia, 18*, 1289 – 1298. DOI: 10.1016/j.egypro.2012.05.146

Saleem, A., Bady, M., Ookawara, S. y Abdel-Rahman, A. (2016). Achieving standard natural ventilation rate of dwellings in a hot-arid climate using solar chimney. *Energy and Buildings, 133*, 360–370. DOI: http://dx.doi.org/10.1016/j.enbuild.2016.10.001

Sánchez, E. (2017). *Optimización de la fachada de doble piel acristalada con ventilación natural. Metodología de diseño para el análisis de la eficiencia energética del sistema*. Tesis doctoral. Universidad Politécnica de Madrid. Recuperado de https://oa.upm.es/46745/1/EDURNE_SANCHEZ_SALCEDO.pdf

Shi, L., Zhang, G., Cheng, X., Guo, Y., Wang, J. y Chew, M. (2016). Developing an empirical model for roof solar chimney based on experimental data from various test rigs. *Building and Environment, 110*, 115-128. DOI: http://dx.doi.org/10.1016/j.buildenv.2016.10.002

Shi, L., Zhang, G., Yang, W., Huang, D., Cheng, X. y Setunge, S. (2018). Determining the influencing factors on the performance of solar chimney in buildings. *Renewable and Sustainable Energy Reviews, 88*, 223-238. DOI: https://doi.org/10.1016/j.rser.2018.02.033

Yusoff, W., Salleh, E., Adam, N., Sapian, A. y Sulaiman, M. (2010). Enhancement of stack ventilation in hot and humid climate using a combination of roof solar collector and vertical stack. *Building and Environment, 45*, 2296-2308. DOI: 10.1016/j.buildenv.2010.04.018

Zhai, X., Dai, Y. y Wang, R. (2005). Comparison of heating and natural ventilation in a solar house induced by two roof solar collectors. *Applied Thermal Engineering, 25*, 741-757. DOI: 10.1016/j.applthermaleng.2004.08.001

Zhang, T., Tan, Y., Yang, H. y Zhang, X. (2016). The application of air layers in building envelopes: A review. *Applied Energy, 165*, 707–734. DOI: https://doi.org/10.1016/j.apenergy.2015.12.108