27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

COB-2023-0238
NUMERICAL ANALYSIS OF A SOLAR CHIMNEY FOR BUILDING VENTILATION

Vinicius Branzani Leite
Thales de Freitas Villas-Bôas
Cyro Albuquerque Neto
Centro Universitário FEI - Av. Humberto de Alencar Castelo Branco, 3972-B, Assunção, São Bernardo do Campo - SP
vini_b_leite@hotmail.com, thalesfvb@gmail.com, cyroan@fei.edu.br

Abstract. *The objective of the present work is the numerical analysis with CFD of the performance of a conventional solar chimney used to ventilate an environment. The application of solar chimney has been studied recently in the scientific literature for heating or cooling environments. Most studies are experimental, and the literature lacks numerical studies on the subject. The transport phenomena inside the chimney are complex, with vortices formed by natural convection and significant influence of external conditions, such as wind, rain, and changes in solar radiation. Solar chimneys can benefit from being a passive and sustainable HVAC solution, which can help reduce energy consumption. The CFD simulation of this device can comprehensively evaluate the phenomena involved. In this work, CFD simulations with the software Ansys CFX were performed and validated with the group's experimental data on a solar chimney 2 m in height, with 50 and 150 mm thickness cavities. The chimney is appropriate to use in a warm climate, with an air inlet from the indoor ambient at the bottom of the solar chimney and the air outlet to the external ambient on the top. Appropriate conditions were used in the simulations for natural convection phenomena, with mesh refinement for representing the boundary layer and large gradients. A far-field was included around the chimney to circulate the air between the outlet and the inlet. The influence of different turbulence models was investigated, which significantly impacts the velocity profile's behavior. It was found that some of the models can accurately represent the experimental data for temperature. Nevertheless, for the air velocity, they can represent the shape of the velocity profile curve but with lower values than the experimental ones. Some analyses were also included to verify the effects of air humidity and hot wall temperature. Both cases have little effect on the behavior of the chimney.*

Keywords: *Natural Convection, Solar Chimney, Passive Ventilation, Heat transfer.*

1. INTRODUCTION

A solar chimney built on the wall of a building can be used for ventilation assistance or heating. The chimney is constructed within a cavity that can be located in the wall or the ceiling. The air inside the cavity is heated by solar energy, causing an upward movement due to natural convection.

Solar chimneys have been studied in recent years, primarily for space heating purposes. In the scientific literature, several reviews (Chan *et al.*, 2010; Khanal and Lei, 2011; Zhai *et al.*, 2011) indicate that further studies are needed to improve efficiency and propose new configurations, which can also be integrated with other systems.

The research group involved in this project has conducted experimental studies on solar chimneys with different configurations for two applications. In one of these studies, the potential application of a solar chimney in shipping containers used as buildings was investigated, utilizing the trapezoidal cavities formed by the folds of the side steel plates and an insulating material (Malta *et al.*, 2014, 2013; Villas-Bôas *et al.*, 2015). The other application focused on proposing a device to remove air from the upper zone of a space, including an additional inlet with a plate that reduces the cross-section of the chimney (Villas-Bôas *et al.*, 2017, 2016; Villas-Bôas, 2019).

Most of the found works are experimental, although there are some numerical studies that assisted in defining and discussing the present work. Gan (1998) conducted a parametric study of a Trombe wall, similar to a solar chimney, with an internal wall capable of retaining energy, commonly used in cold environments, using a CFD tool. The developed code can be used to predict the phenomena caused by natural ventilation. In another work, Gan (2010) presented a study on the computational impact of simulations related to natural ventilation in buildings. The author points out that numerical studies on natural ventilation have mainly focused on analyzing only the internal cavity of the chimney, often assuming that the inlet and outlet have constant pressures, which leads to a nearly constant velocity profile at the inlets and results in significant errors. Moshfegh and Sandberg (1998) numerically and experimentally analyzed the flow and heat transfer in a duct behind photovoltaic panels. The analysis revealed the importance of radiation in the heat exchange between the cavity walls. For heat fluxes greater than 200 W/m^2, approximately 30% of the heat is transferred to the other non-heated wall and then dissipated by the air through convection. More recently, Kong *et al.* (2020) studied the optimal inclination angle of solar chimneys with CFD, showing recirculation formations within the flow. Other applications recently studied with CFD are the application of phase change materials in cold places, managing to maintain ventilation for longer after sunset (Jiménez-Xamán *et al.*, 2019).

This work aimed to improve the studies conducted by Villas-Bôas (2019), looking for a numerical validation of the conducted experiments in different configurations. The study was conducted using a cavity measuring 0.4 m wide by 2 m height, varying its thickness at 0.050 m and 0.150 m, and a configuration with two air inlets, one at floor level and another 1 m above it.

2. METHODOLOGY

2.1 Geometry and Mesh

The two-dimensional geometries were modeled based on the longitudinal section of the chimney studied by Villas-Bôas (2019), one measuring 0.050x2 m and the other 0.150x2 m. The control volume composed of this chimney has a far field that is ten times the size of its cavity thickness, as suggested by Gan (2010). The mesh discretization is divided into two modes: on the relevant surfaces, i.e., inside the cavity and around the hot wall, a structured mesh was created, while in the other regions, an unstructured mesh was used. In the cavity inlet and outlet regions, it was applied a gradual refinement of the element size. An element size validation was performed to stabilize the obtained results, considering an error of 2%. The refinement aimed to comply with the conditions of the logarithmic law of the wall, as suggested by the Ansys manual (Ansys, 2011), where the height of the first element near the surface should have a y+ value close to 1 in order to capture the effects of the sub-viscous layer of the flow. The final meshes can be observed in Figures 1 and 2, and the validation results are shown in Table 1.

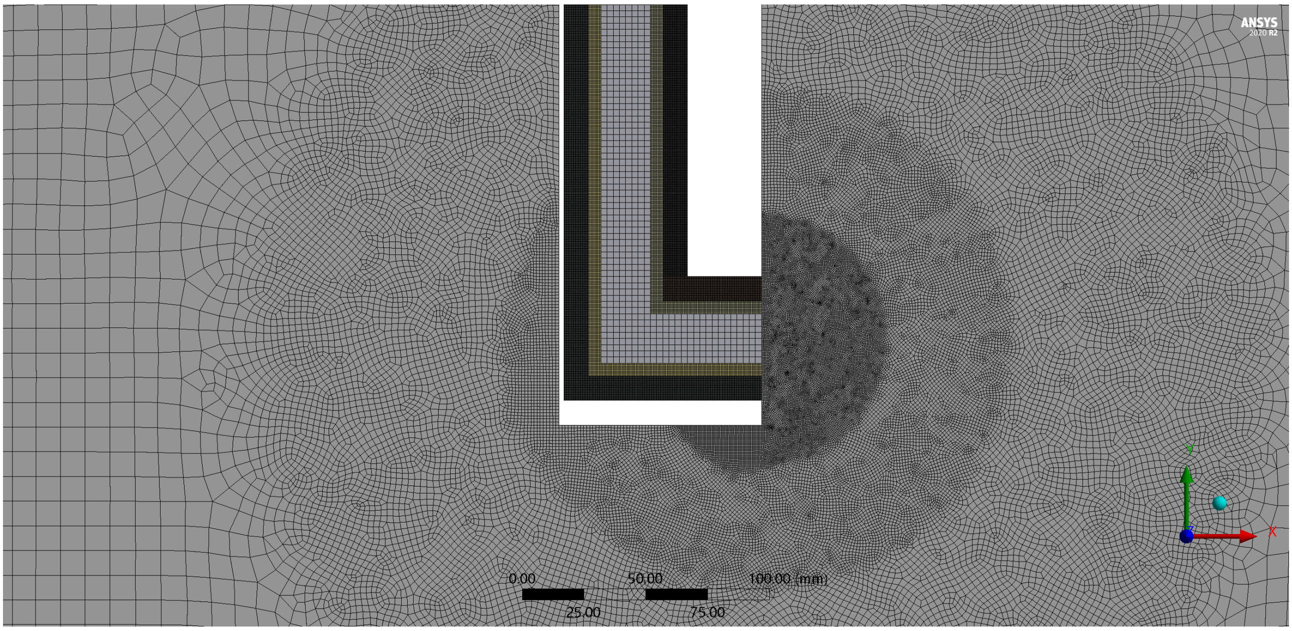

Figure 1. Chimney detailed mesh.

Table 1. Mesh validation. The relative deviation is for the average velocity at the height of 1 m of the chimney. Temperature relative deviations present negligible values compared to velocity.

		Number of elements	**Relative deviation %**
50mm	Simulation 1	202043	
	Simulation 2	281735	0.216%
	Simulation 3	593543	0.294%
150mm	Simulation 1	364142	
	Simulation 2	490982	0.255%
	Simulation 3	921974	1.532%

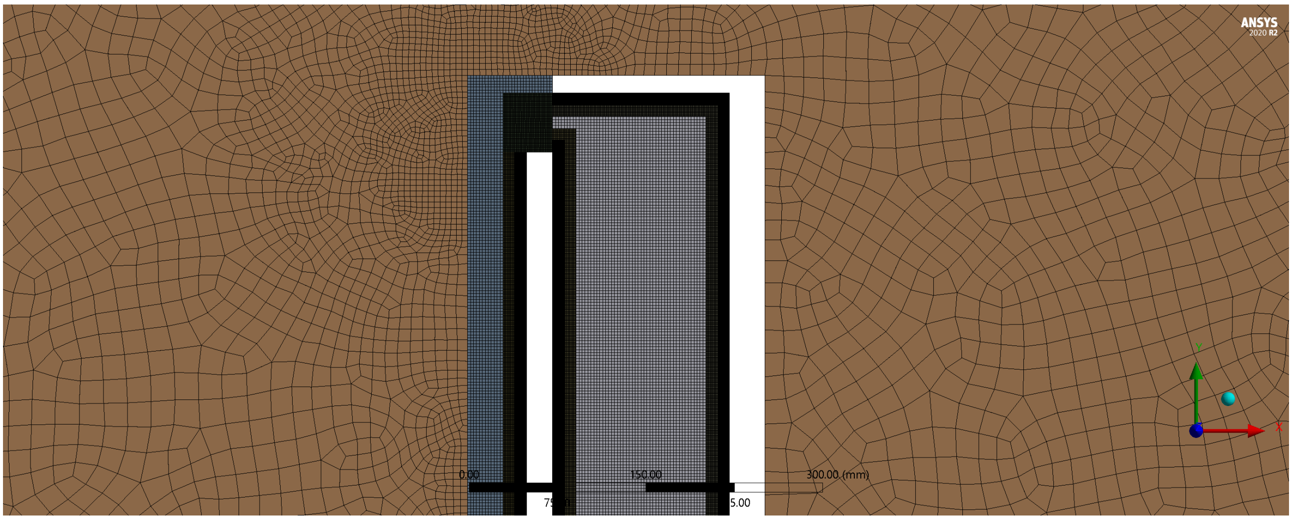

Figure 2. Chimney mesh considering heat from external side.

2.2 Preprocessing

In this section, the boundary conditions and criteria used in the simulation are discussed. The properties considered are the same as those in the conducted experiment. Thus, the atmospheric pressure is 93085.7 Pa, with a temperature of 20 °C. The "cold" walls, i.e., those without heat-generating resistance, are considered to have a temperature of 20 °C. The walls inside the cavity, except for the "hot" wall, are considered adiabatic. Initially, it is assumed that the "hot" wall temperature is 50 °C, but this value was also adopted as 80 °C, as will be discussed later. The flow is considered steady and two-dimensional. The influence of radiation is also taken into account, using a discrete transfer model. The emissivity values of the walls are obtained from the composite materials of the chimney experiment, which can be seen in the Table 2. The turbulence models chosen for analysis are the two-equation models, RNG k-ε, k-ω, k-ω SST, as well as the six-equation SSG Reynolds Stress model. The two-equation models were chosen due to their practicality and stability, commonly used in literature and industry. On the other hand, the SSG model, despite being more unstable, was chosen as suggested in the CFX manual (Ansys, 2011) for flows with buoyancy influence.

Table 2. Wall emissivities.

Wall	**Emissivity**
Hot	0.09
Adiabatic	0.60
Cold	0.85
Farfield	1

The convergence criteria are met when the root mean square residuals for mass, momentum, and energy are below 10^{-5}. First-order upwind advection scheme is used for mesh validation, while the second-order scheme is used for the actual analysis.

3. RESULTS AND DISCUSSION

3.1 Hot wall temperature

Initially, the temperature measured during the experiment is 50 °C. However, during the simulations, a discrepancy between the simulated curves and those from the experiment is observed. It is suspected that the actual temperature on the wall is higher than the one measured by the thermocouple. Therefore, the temperature is corrected to 80 °C, which shows good agreement with the curve obtained in the experiment, as shown in figure 3.

3.2 Validation

The obtained results were compared with the experimental data and are shown in Figures 4 and 5. It can be observed that, despite some similarities in the velocity profile shape, there is a discrepancy between the numerical and experimental results. Thus, two hypotheses are investigated in this work: changes in absolute air humidity and heat generated from both sides of “hot” wall of the chimney.

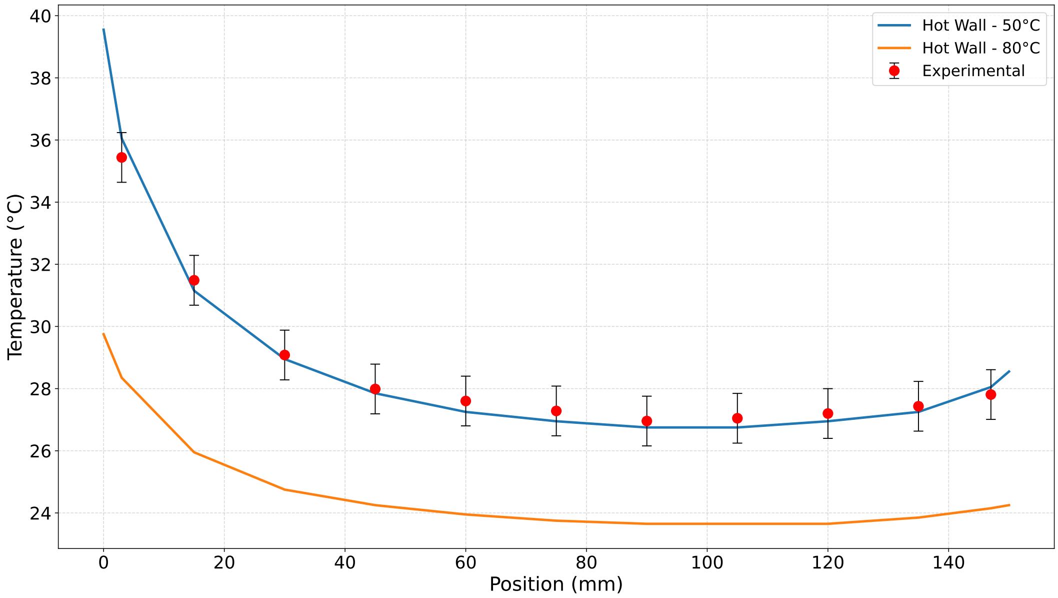

Figure 3. Effect of hot wall temperature on the air temperature profile.

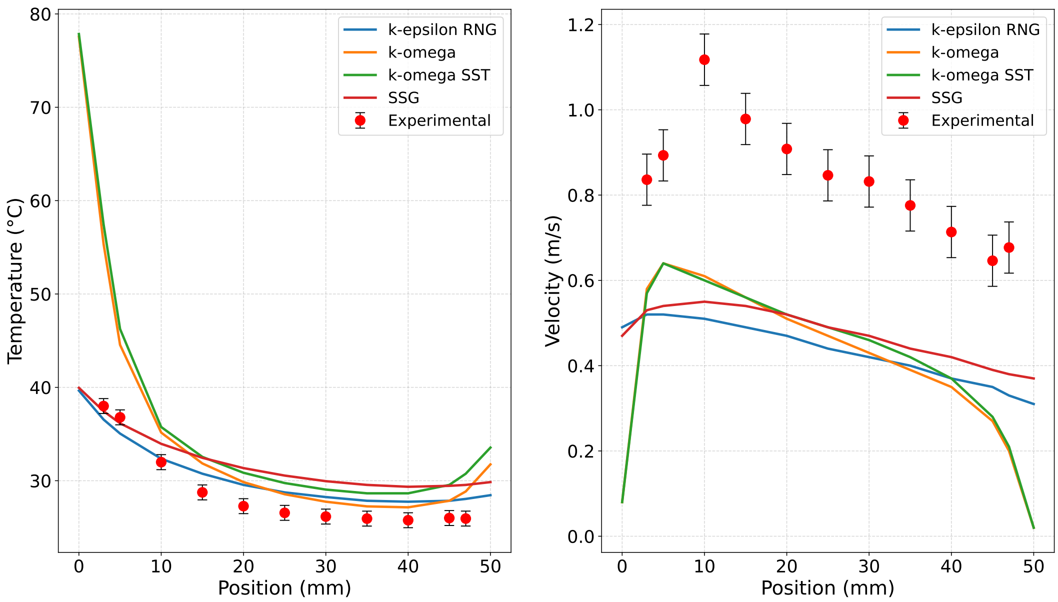

Figure 4. Temperature profiles (right) and velocity profiles (left) for 50mm width chimney with different turbulence models.

Although k-ω-based models are more robust at low Reynolds numbers, these models exhibit a significant temperature decay near the wall. Therefore, the RNG k-ε model proves to be the best-represented model in this situation. To evaluate the mentioned hypotheses, the RNG k-ε model is used due to its better behavior.

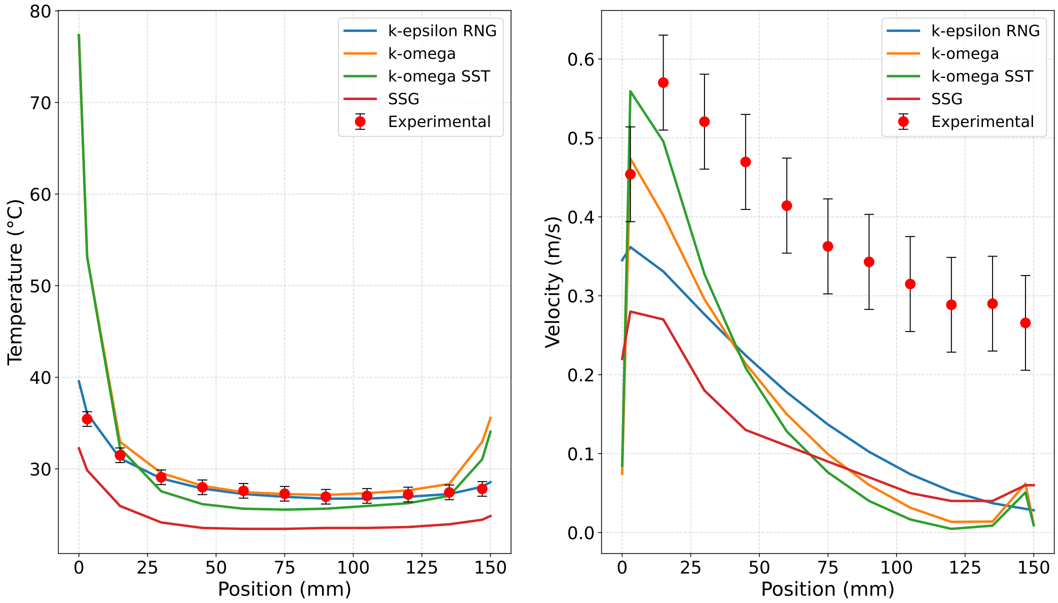

Figure 5. Temperature profile (right) and velocity profile (left) for 150mm width chimney

3.3 Influence of air humidity

The first hypothesis studied was the influence of absolute air humidity at the time of the experiment. It is known that with an increase in air humidity, more energy transfer between the flow should occur since the specific heat capacity (c_p) increases. As the air humidity was not measured in the experiment, it was decided to calculate the variation of (c_p) from 10% to 80% relative humidity, according to psychrometric relations:

$$c_p = c_{p,a} + c_{p,v}\,\omega \tag{1}$$

where:

- $c_{p,a}$: specific heat capacity of dry air, 1.005 kJ/kgK;
- $c_{p,v}$: specific heat capacity of vapor, 1.82 kJ/kgK;
- ω: absolute humidity in kg of vapor/kg of dry air.

The values are obtained from Table 3, acquired at the reference temperature of 30 °C. It can be observed that the variation in thermal capacity with respect to absolute humidity is very small, and it is concluded that there would be no significant change in the results. A high humidity of 80% would result in only 1.75% of vapor mass in the air mass.

Table 3. Specific heat capacity variation.

Relative humidity	**Absolute humidity** (kg_{vapor}/kg_{arseco})	**Specific heat capacity** (kJ/kgK)
10%	0.0021	1.009
20%	0.0043	1.013
30%	0.0065	1.017
40%	0.0086	1.021
50%	0.0108	1.025
60%	0.0131	1.029
70%	0.0153	1.033
80%	0.0175	1.037

3.4 Influence of the external wall

The second hypothesis stems from the fact that the heat-generating resistance was transferred both inside and outside the chimney, which could cause the externally generated flow to assist the air flow at the outlet, dragging the air from the internal flow upward. The results of the velocity profile, as well as the flow details, can be seen in Figures 6 and 7.

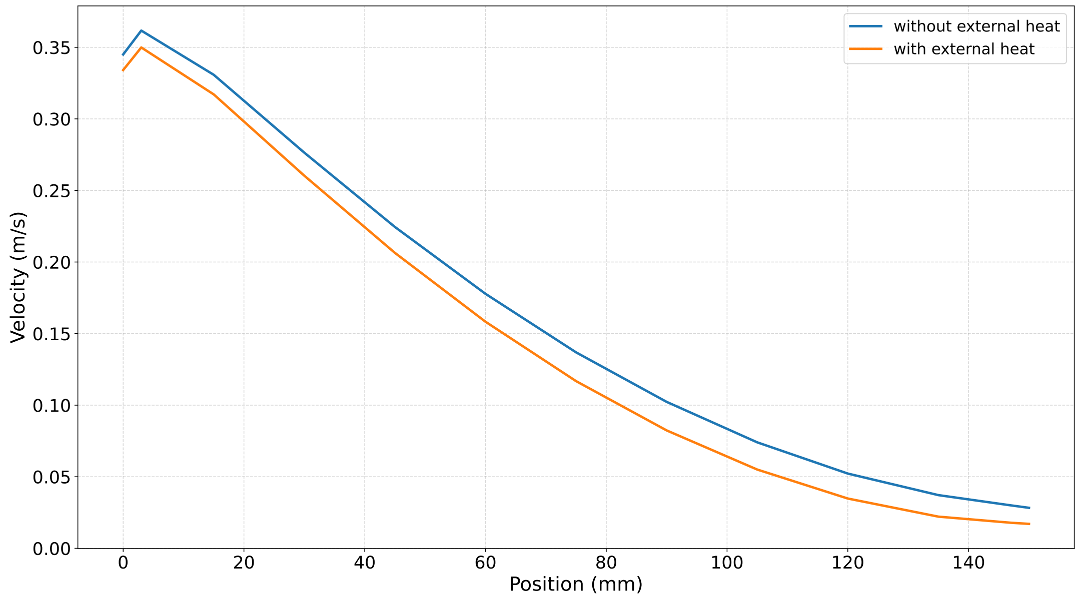

Figure 6. Comparison between chimney flow with external heat influence and without influence.

A slight impairment in the velocity distribution inside the chimney is observed. This is due to the fact that the external flow hinders the outlet instead of contributing to it. As shown in Figure 8, a narrowing of the flow at the outlet can be observed, which hampers the free movement of air within the chimney.

4. CONCLUSION

The simulation in this work consisted of using an experiment conducted on a solar chimney for the validation of a numerical study. The care to be taken when determining the boundary conditions is evident, considering the influence of radiation, the exact values of the measurements, and accounting for losses and external influences during the experiment. Despite being a two-dimensional and steady flow model, the computational capacity required was relatively high for the radiation simulation, with the initial calculation time taking more than half of the total simulation time. Regarding turbulence models, the renormalized k-ε model provided the best response for this case, with temperature points very close to the experimental data. Further analysis is still needed to understand why the simulation exhibits lower velocity curves compared to those obtained in the experiment. It can be discussed that the vortices generated by an upward natural convection flow are essentially three-dimensional and transient, as well as the interaction between an external flow and the flow at the chimney outlet. Therefore, as future work, a three-dimensional and/or transient study using models such as LES is suggested, where capturing these low-frequency vortices can be an advantage in representing this type of flow.

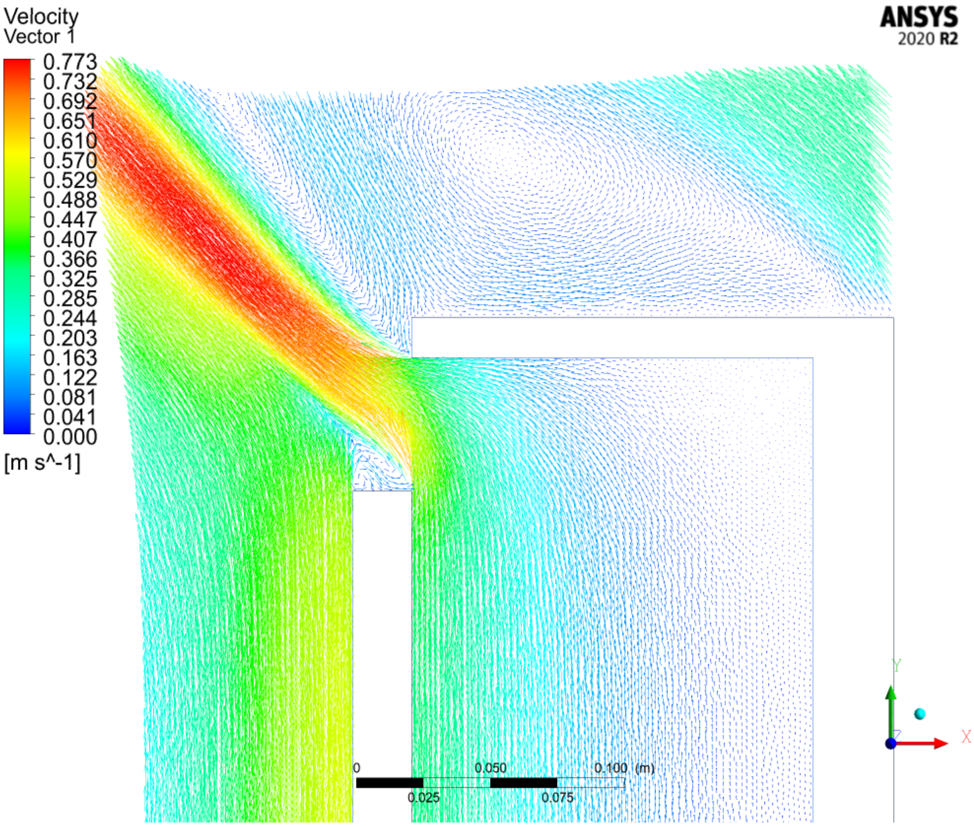

Figure 7. Velocity vector distribution at chimney outlet.

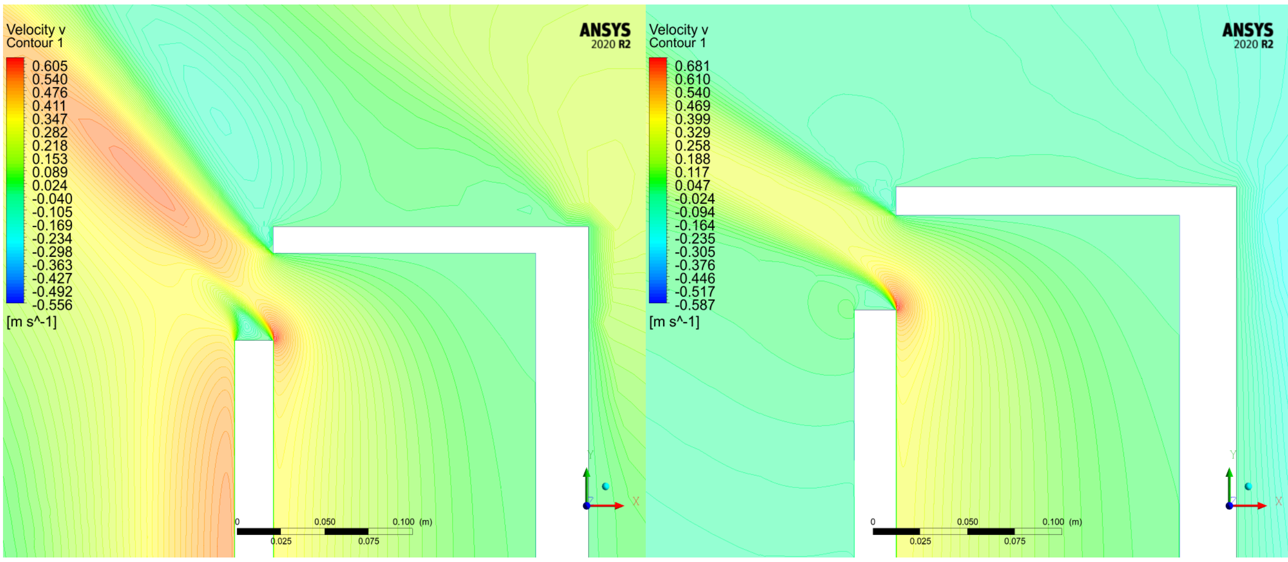

Figure 8. Chimney velocity contour with external influence (left) and without external influence (right)

5. REFERENCES

Ansys, 2011. "ANSYS CFX Solver Modeling Guide".

Chan, H.Y., Riffat, S.B. and Zhu, J., 2010. "Review of passive solar heating and cooling technologies". *Renewable and Sustainable Energy Reviews*, Vol. 14, No. 2, pp. 781–789.

Gan, G., 1998. "A parametric study of trombe walls for passive cooling of buildings". *Energy and Buildings*, Vol. 27, No. 1, pp. 37–43.

Gan, G., 2010. "Impact of computational domain on the prediction of buoyancy-driven ventilation cooling". *Building and Environment*, Vol. 45, No. 5, pp. 1173–1183.

Jiménez-Xamán, C., Xamán, J., Moraga, N.O., Hernández-Pérez, I., Zavala-Guillén, I., Arce, J. and Jiménez, M., 2019. "Solar chimneys with a phase change material for buildings: An overview using cfd and global energy balance". *Energy and Buildings*, Vol. 186, pp. 384–404.

Khanal, R. and Lei, C., 2011. "Solar chimney-a passive strategy for natural ventilation". *Energy and Buildings*, Vol. 43, No. 8, pp. 1811–1819.

Kong, J., Niu, J. and Lei, C., 2020. "A cfd based approach for determining the optimum inclination angle of a roof-top solar chimney for building ventilation". *Solar Energy*, Vol. 198, pp. 555–569.

Malta, F.P., Gouveia, J.L.F. and Albuquerque, C., 2013. "Investigation of natural convection in solar chimney with trapezoidal section to improve comfort inside shipping containers". In *Proceedings of the 22nd International Congress of Mechanical Engineering*. ABCM, Ribeirão Preto.

Malta, F.P., Gouveia, J.L.F. and Albuquerque, C., 2014. "A study of a solar chimney to improve comfort inside shipping containers". In *Proceedings of the 27th International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems*. Abo Akademi University, Turku, Finland.

Moshfegh, B. and Sandberg, M., 1998. "Flow and heat transfer in the air gap behind photovoltaic panels". *Renewable and Sustainable Energy Reviews*, Vol. 2, No. 3, pp. 287–301.

Villas-Bôas, T.F., 2019. *Desenvolvimento de uma chaminé solar com dupla entrada para remoção de ar da zona superior e inferior de ambientes internos*. Master's thesis, Engenharia Mecânica, Centro Universitário FEI, São Paulo.

Villas-Bôas, T.F., Malta, F.P. and Albuquerque, C., 2017. "Analysis of a solar chimney with double inlet by means of venturi effect". In *Proceedings of the 24th ABCM International Congress of Mechanical Engineering*. ABCM, Curitiba.

Villas-Bôas, T.F., Malta, F.P., Gouveia, J.L.F. and Albuquerque, C., 2015. "Study of different configurations of solar chimney to improve comfort inside shipping containers". In *Proceedings of the 23nd International Congress of Mechanical Engineering*. ABCM, Rio de Janeiro.

Villas-Bôas, T.F., Malta, F.P., Gouveia, J.L.F. and Albuquerque, C., 2016. "Study of a wall solar chimney to remove air from the top of an indoor environment". In *Proceedings of the 16th Brazilian Congress of Thermal Sciences and Engineering*. ABCM, Vitória.

Zhai, X.Q., Song, Z.P. and Wang, R.Z., 2011. "A review for the applications of solar chimneys in buildings". *Renewable and Sustainable Energy Reviews*, Vol. 15, No. 8, pp. 3757–3767.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.