Journal of Theoretical and Applied Mechanics, Sofia, Vol.55 (2025) pp. 91-104

FLUID MECHANICS

NUMERICAL STUDY OF WIND-ASSISTED VENTILATION IN A SOLAR CHIMNEY

OMAR BELHADJ[1], HAMIDOU BENZENINE[1,2]*
SALIM NEWAZ KAZI[3], RACHID SAIM[1]

[1] *Laboratory of Energetic and Applied Thermal ETAP,*
Department of Mechanical Engineering, Faculty of Technology,
B.P 230, University of Tlemcen, 13000, Algeria
[2] *Department of Mechanical Engineering, Faculty of Science and Technology,*
University Belhadj Bouchaib, BP 284 RP, Ain Témouchent, 46000, Algeria
[3] *Department of Mechanical Engineering, Faculty of Engineering,*
University Malaya, 50603 Kuala Lumpur, Malaysia

[Received: 4 October 2024. Accepted: 20 March 2025]

doi: https://doi.org/10.55787/jtams.25.55.1.091

ABSTRACT: A numerical analysis of a two-dimensional turbulent air flow in a solar chimney, with a Venturi device at the outlet, was conducted. The Venturi device creates a pressure drop, significantly enhancing the air evacuation at the outlet. Simulations were performed using Ansys Fluent, employing the finite volume method to determine and analyze the dynamic and thermal fields under the influence of wind velocity.

The validation of the numerical model with the experimental results shows good agreement. The results demonstrate a linear and direct proportionality between the increase in wind speed and the ventilation flow rate at the outlet, keeping other factors constant. The system also showed the presence of ventilation even in the absence of wind.

KEY WORDS: Energy security, energy efficiency in buildings, natural ventilation, solar chimney, wind effect.

1 INTRODUCTION

Sustainable systems in buildings play a crucial role in enhancing energy security. They achieve this by reducing energy consumption.In the future, the energy demand may double or multiply or even triple as the population grows and developing countries expand their economies.

Today the building sector accounts for about 35.3% of the global final energy consumption [1]. Given this, designing buildings with very little energy consumption has become increasingly important. Consequently, the use of passive techniques is emerging as a good strategy for energy efficiency in buildings.

---

*Corresponding author e-mail: hamidou.benzenine@univ-temouchent.edu.dz

Generally, the energy requirement in buildings is affected by Heating, Ventilation, and Air Conditioning (HVAC) systems, which consume more than 60% of the total energy in Buildings [2]. This has led to an increase the use of sustainable building systems like, wind catchers, Passive cooling, heating systems, Trombe walls, and Solar chimneys as techniques to reduce building energy consumption.

Passive systems are increasingly proposed for natural ventilation, as they can replace mechanical ventilation systems; because of their potential benefits in terms of: operational cost, energy requirement and emissions reduction. Natural ventilation offers cooling energy saving by 10% and fan power savings about 15% of annual energy consumption when climatic and operational conditions are suitable [3].

Solar chimney is a simple and efficient tool to improve natural ventilation. It transforms solar irradiation into kinetic energy to extract air from the interior space from the external ambient to provide comfort. It can replace mechanical ventilation systems because of their potential benefits in terms of operational cost, energy requirement and carbon dioxide emission.

A solar chimney system creates a difference in the density of air between the inside and outside causing forces called buoyancy forces.

Many studies carried out to understand solar chimney's behavior, conventional and solar chimney are compared by Afonso et al. [4]; they observed 10% to 22% increase in efficiency by solar assistance. The results of C. Jiménez-Xamán et al. [5] showed that in a room with an integrated solar chimney, the ventilation rate is improved by $8--45\%$ in summer and by $1.16--24.89\%$ in winter. Xinyu Zha et al. [6] examine the performance of a full-scale vertical solar chimney in a real building in Eastern China, it illustrates that the solar chimney is an effective technique to save energy for residential buildings in transition seasons in areas with hot summer and cold winter; the energy saving rate is around 14.5%.

The main goal of designing a solar chimney is to optimize its performance with the lowest cost. For this challenge, it is important to identify the factors influencing the efficiency as key factors to improve it. Solar chimneys receive considerable attention by researchers [7–13] in order to improve their performances; the main influencing factors are installation and position, geometry, material usage and environment (location and meteorological conditions).

The wind can have a significant impact on building ventilation. It creates a fascinating phenomenon known as pressure drop between the windward side and the leeward side of a building. This pressure difference plays a crucial role in ensuring proper airflow and ventilation inside buildings known as wind-driven ventilation. By understanding how wind affects ventilation, we can optimize our buildings to maximize energy efficiency and improve occupant well-being.

External wind shows significant influence on the solar chimney performance [14].

Due to wind, high-pressure is generated from one side of the building pushes air into the building; in this situation the wind improve significantly the ventilation while the solar chimney is installed on the low-pressure side.

A wind-induced channel was designed and studied by Ning Gao et al. [15] to enhance indoor natural ventilation under a combination of wind and solar energy action; it is found that the ventilation rate and indoor air quality are influenced by the external wind speed.

In some configurations, other arrangements are included at the exhaust of the chimney in order to enhance the ventilation efficiency. For example, article [16] shows wind-assisted solar chimneys used to drive natural ventilation through the classrooms of a school in Damascus; the chimneys are designed to use wind to create negative pressure at the top of the chimney which improves the stack air movement inside the chimney. E. Mahnoosh and M. Abbas [17] designed a zero-energy passive system to ventilate the building and provide comfortable conditions; the solar chimney is assisted by the wind catcher to improve the ventilation system. Haw LC et al. [18] explored the application of a wind-induced natural ventilation tower viability in building under the hot and humid climatic conditions with a Venturi-shaped roof geometry; this design offers 57 air changes per hour (ACH) at an external wind velocity of 0.1 m/s when, with only the cross ventilation, the air change is only 7 air changes per hour (ACH).

The present study aims to combine solar and wind impact to improve natural ventilation considering the random nature of wind. It demonstrate how to boost the ventilation system using the wind impact on the solar chimney exhaust for any wind direction and speed. The design should have advantageous contribution of the wind because the wind can have positive or negative impact on the top of the solar chimney depending on the direction.

2 DESCRIPTION

When the wind blows, with an appropriate geometry at the chimney exhaust, such as the venturi, it creates a pressure drop that can boost the flow through the chimney and provide better ventilation. In this situation, the wind acts at the exit of the chimney. The pressure difference generated by the wind near the outlet area of the chimney structure is added to the buoyancy effect; consequently, the driving force for the ventilation is higher.

Moreover, the adopted system can serve the following purposes:

- Improve the solar chimney performance
- Extend the ventilation process during night-time or cloudy days

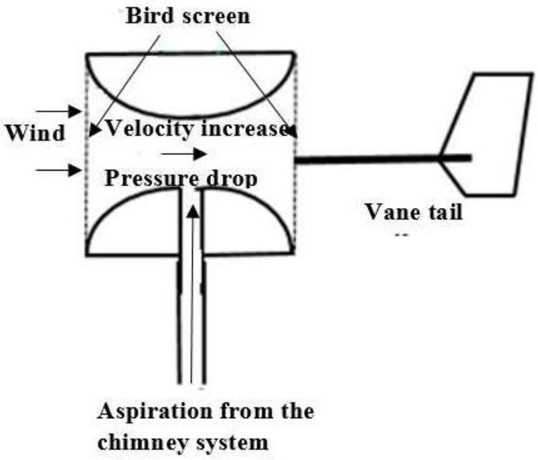


Fig. 1. Chimney cap arrangement for wind assistance.

- serve as a chimney cap keeping out rainwater, snow and debris going down the chimney
- Prevent animals mainly birds from building nests inside the chimney
- Prevent undesirable wind interference

In order to catch advantageous contribution of wind and prevent undesirable wind interference, the chimney top should be strategically positioned. A vane tail is designed to allow the device to catche the wind direction (Fig. 1). When mounted on the system, the vane rotates under the influence of the wind such that its center is to leeward and directs the assembly into the wind direction.

3 SIMULATION

3.1 MODEL GEOMETRY AND BOUNDARY CONDITIONS

A 2D computational domain is used. Figure 2 shows the structure of chimney with solar and wind assistance, composed of two main parts: the ventilation channel and the wind channel. The ventilation channel consists of the absorber and glazing installed on the south wall in order to catch solar advantages and conduct the extracted air from the interior space to the exterior. The second part is the wind channel is installed on top of the ventilation channel. It features a Venturi-shaped cap designed to capture the advantages of wind.

Boundary conditions are the following:

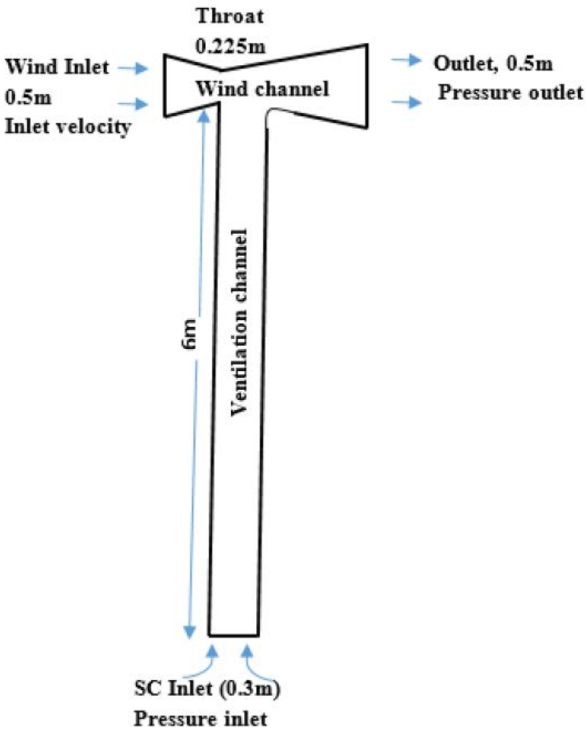


Fig. 2. Geometry and boundary conditions.

Inlets

At the inlet section of the solar chimney, a pressure inlet boundary condition is set to define the fluid pressure, pressure inlet boundary conditions is used since the flow rate and velocity are not known. The total (gauge) pressure is set to zero at the inlet, and the inflow is normal to the inlet. The incoming air is at the reference temperature.

In contrast, at the wind channel inlet, the incoming velocity value is imposed normal to the inlet as the wind velocity.

Outlet

At the outlet area, we consider the pressure outlet boundary condition; it is assumed that the stream wise variations of all velocity components and temperature are negligible. In case of a backflow at the exit, its velocity is supposed to be normal to the exit boundary and the backflow temperature is at the reference temperature.

Walls

No slip boundary conditions are adopted on solid boundaries. A constant heat flux is imposed at the absorber wall. All other walls are adiabatic.

3.2 GOVERNING EQUATIONS AND ASSUMPTIONS

The numerical simulation is done using a 2D computational fluid dynamics model. For the natural ventilation prediction, the fluid is Newtonian; the flow is considered steady, incompressible and turbulent. The standard k-ε model with a standard wall functions is used. Material properties are independent of temperature except for the density in the buoyancy term; density variation caused by temperature rise is expressed using Boussinesq approximation.

The equations represent the conservation principles of mass, momentum, and energy of the flow:

Continuity

$$\frac{\partial U_i}{\partial x_i} = 0\,. \tag{1}$$

Momentum

The expression for conservation of momentum (Navier-Stokes equations) is

$$\frac{\partial}{\partial x_j}\left(U_i U_j\right) = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\nu+\nu_t)\left(\frac{\partial U_i}{\partial x_j}\right)\right] + \underbrace{[1-\beta(T-T_0)g]}_{\text{buoyancy}}\,. \tag{2}$$

Energy

$$\frac{\partial\left(U_j T\right)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\Gamma\frac{\partial T}{\partial x_j}\right)\,, \qquad \Gamma = \frac{\nu}{P_r} + \frac{\nu_t}{\sigma_t}\,. \tag{3}$$

k-equation of turbulent kinetic energy of turbulence model:

$$\frac{\partial}{\partial x_i}\left(kU_i\right) = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \varepsilon - Y_M + S_k\,. \tag{4}$$

ε-equation of dissipation rate of turbulent model

$$\frac{\partial}{\partial x_i}\left(\varepsilon U_i\right) = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\frac{\varepsilon^2}{k} + S_\varepsilon\,, \tag{5}$$

where U_i is the component of average velocity in x_i direction, [m/s]; $(i = 1, 2)$ so X and Y are coordinate axis in 2D. P is the average pressure, [Pa]; ρ is the air density, [kg/m^3]; T, T_0 are the average temperature and the reference point temperature, respectively, [°K]; Γ is the generalized diffusion coefficient; k is the turbulence kinetic energy and ε its rate of dissipation; ν and ν_t are the laminar and the turbulent (or eddy) viscosity.

To evaluate G_k in a manner consistent with the Boussinesq hypothesis:

$$G_k = \nu_t S^2 \,,$$

where S is the modulus of the mean rate-of-strain tensor, defined as: $S \equiv \sqrt{2 S_{ij} S_{ij}}$.

The model constants have the following values:

$$C_{1\varepsilon} = 1.44, \qquad C_{2\varepsilon} = 1.92, \qquad C_\mu = 0.09, \qquad \sigma_k = 1.0, \qquad \sigma_\varepsilon = 1.3$$

G_b is the generation of turbulence due to buoyancy defined by

$$G_b = \frac{1}{\rho} \beta g_i \frac{\nu_t}{Pr_t} \frac{\partial T}{\partial x_i} \,.$$

For ideal gases $G_b = -\frac{1}{\rho} g_i \frac{\nu_t}{Pr_t} \frac{\partial \rho}{\partial x_i}$; Pr_t is the turbulent Prandtl number, g_i is the component of the gravitational vector in i-th direction [m/s^2]; β is the thermal expansion coefficient

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p \,.$$

The buoyancy effects on ε are neglected simply by setting G_b to zero in the transport equation for ε. However, the buoyancy effect on ε can be included. In this case, the value of G_b is given considering air as an ideal gas.

$C_{3\varepsilon}$ determines the degree to which ε is affected by the buoyancy, it is calculated by $C_{3\varepsilon} = \tanh |v/u|$, v and u are the components of velocity parallel and perpendicular to the gravitational direction respectively.

So $C_{3\varepsilon}$ becomes unity when the flow is parallel to the gravity direction and zero (0) when the flow is perpendicular to the gravitational vector. Y_M indicates the compressibility effect, which is neglected in modelling incompressible flows. S_k and S_ε are source terms.

3.3 MESH AND SENSITIVITY ANALYSIS

A surface meshing method with quadrilateral mesh and refinement near walls is used. The accuracy of calculation is dependent on the number of cells used to represent the domain. If few cells are used small scale may not be resolved and if too many cells are used the computational cost will be incredibly high in terms of time and memory. A mesh sensitivity study was carried out to have the best compromise between accuracy and time. The number of cells is optimized by means of a grid independence test. The key value is the average velocity of the ventilation air (at the inlet of the solar chimney). Figure 3 shows that there is no significant change in average velocity beyond the set point.

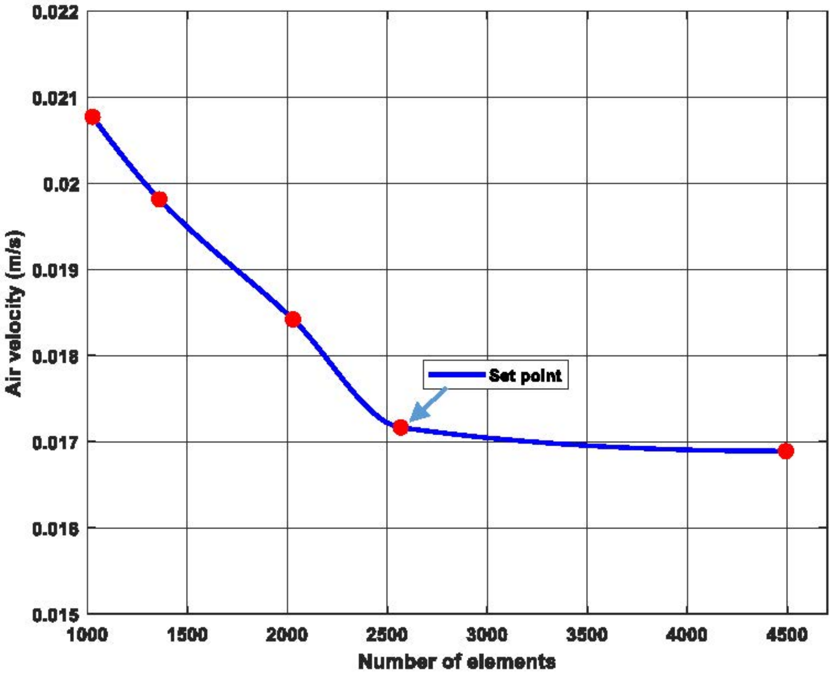


Fig. 3. Mesh sensitivity analysis.

3.4 VALIDATION

CFD process solves systems of non-linear partial differential equations in discretized form on meshes of finite control volumes that cover the domain of interest and its boundaries. This process gives rise to uncertainty and errors.

Errors and uncertainty are inevitable aspects in CFD; therefore, it becomes necessary to validate the simulation tool in order to quantify the level of confidence in its results. For this reason, CFD and previous experimental results [19] were compared; key parameters selected for the comparison are the average velocity and temperature of the ventilating air.

Figure 4 indicates that the results obtained numerically show a smaller difference compared to the experimental results. The values indicate that the model underestimates the temperature behavior by 7.5% to 12%. Regarding the velocity, the model sometimes underestimates and sometimes overestimate the value during the test period. Hence, numerical results show a very good agreement with the experimental results.

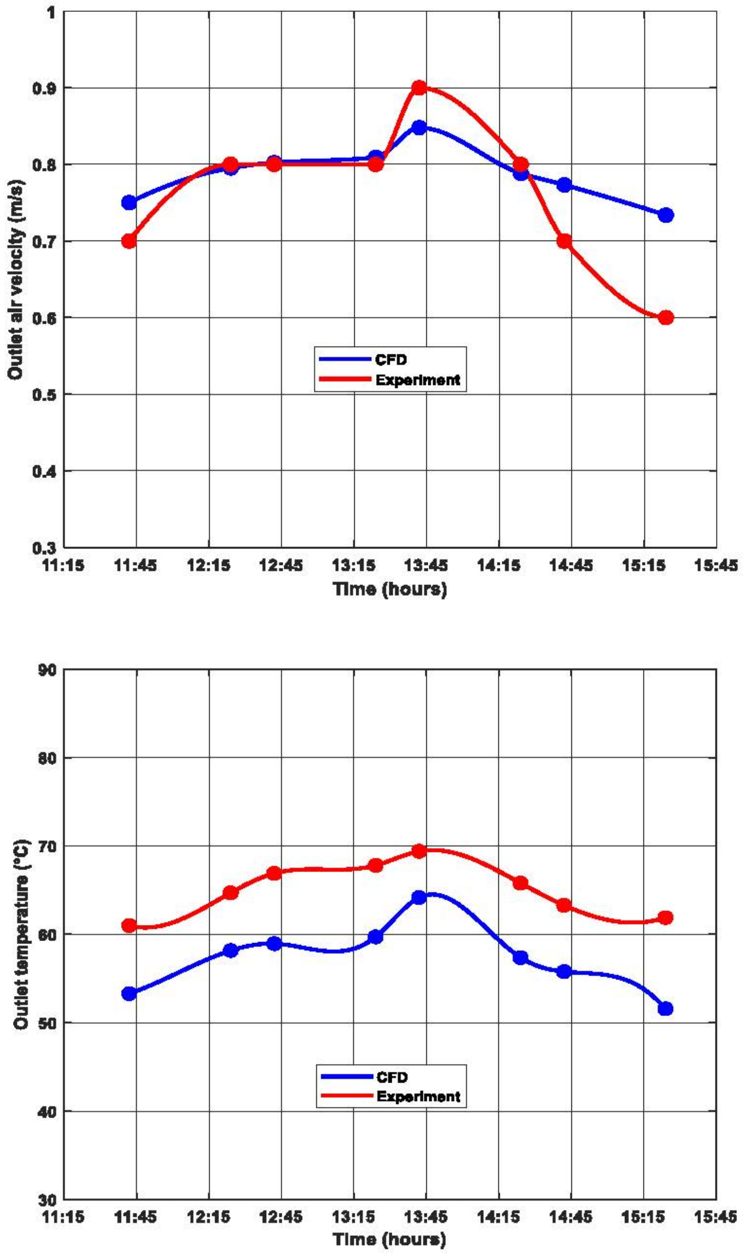


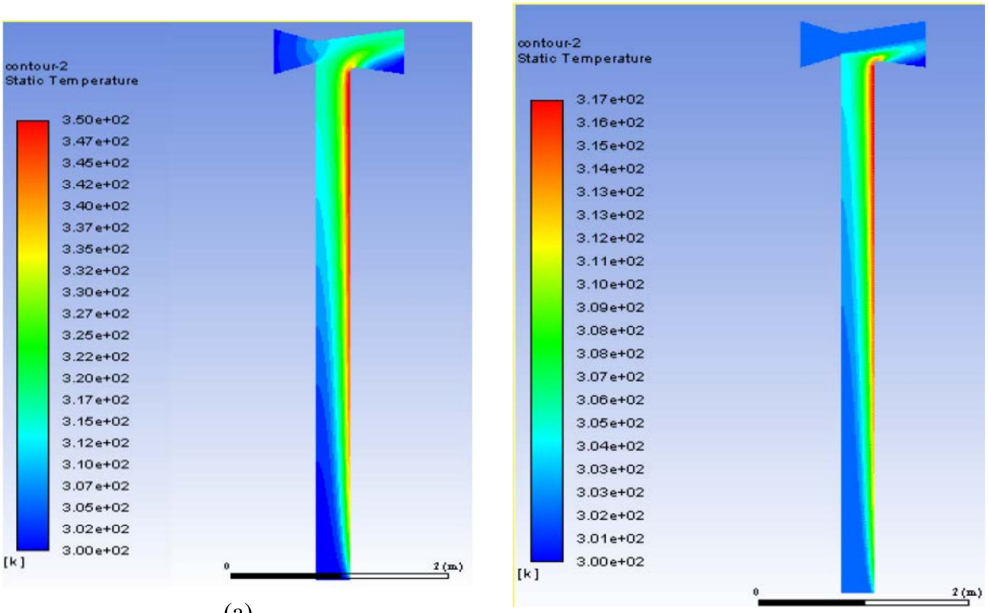
Fig. 4. Experimental and numerical results comparison for the outlet air velocity and temperature.

4 RESULTS AND DISCUSSION

A solar chimney operates on the principle of buoyancy-driven natural convection. The sun's energy heats the air trapped in the channel, causing it to become warmer and less dense, as shown in Fig. 5. This warm and low-density air is lighter than the surrounding cooler air, creating a pressure difference between the base of the chimney and the outside environment.

As the heated air rises, it flows upward through the chimney. The air's buoyancy causes it to move from the high-pressure area at the base (cooler ambient air) to the low-pressure area at the top (warmer rising air). As the air ascends, it maintains a pressure gradient along the height of the chimney, with higher pressure at the base and lower pressure at the top (Fig. 6).

In the convergent-divergent (CD) channel, the pressure contours vary along the length of the channel due to changes in the channel's geometry and the fluid's behavior. The pressure contours in a CD channel depend on the flow regime, fluid properties, and the specific design of the channel. Here, since the pressure distribution in a solar chimney is the key factor that drives the airflow within the chimney, the CD channel is used to create a pressure drop in the SC exhaust using the wind power. Figure 6 shows the pressure distribution in the SC and the CD channel; it is



(a) (b)

Fig. 5. Temperature contours in the CFD domain: (a) $V_{\text{wind}} = 0$ m/s; (b) $V_{\text{wind}} = 0.1$ m/s.

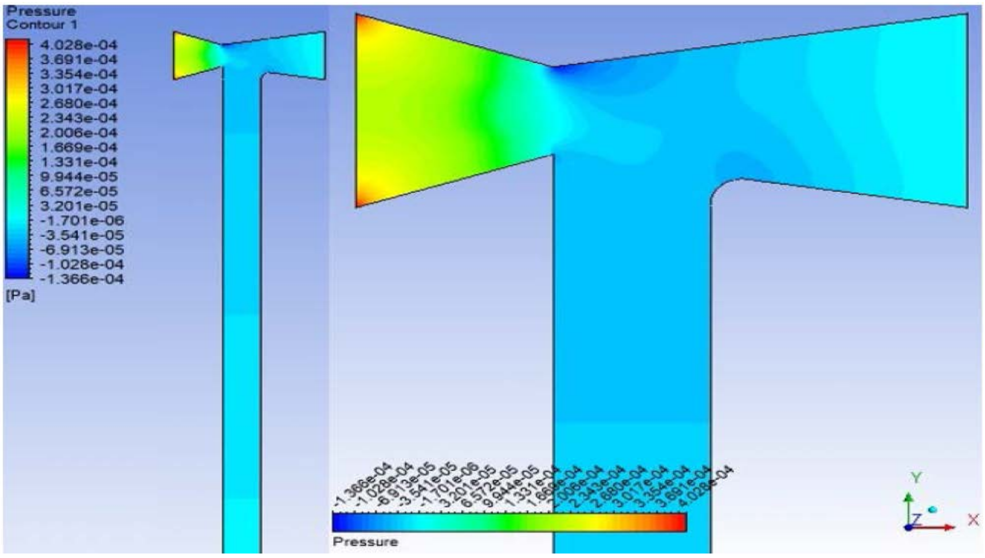


Fig. 6. Pressure distribution in the SC and wind channel CD: $V_{\text{wind}} = 0.1$ m/s.

noticeable that the lower pressure area is located at the intersection of the CD and the SC channels.

In the convergent section of the channel, the flow is accelerated as the cross-sectional area of the channel decreases; according to Bernoulli's principle, as the fluid velocity increases, the pressure decreases. The pressure contours in the convergent section typically show a gradual decrease in pressure from the inlet towards the throat.

The throat is the narrowest part of the channel. At this point, the flow reaches its highest velocity (Fig. 7), and the pressure is at its lowest value. The pressure contours at the throat show the lowest pressure value along the channel (Fig. 6). This decrease in pressure is a result of the fluid's acceleration through the converging section providing a favorable pressure gradient inside the SC and an increase in flow rate driven by the SC.

As the fluid exits the throat and enters the divergent section, it begins to decelerate. The pressure contours in the divergent section typically show an increase from the throat to the exit.

It is important to note that the actual pressure contours can be influenced by factors such as the shape of the channel, the angle of divergence and the presence of shocks.

The velocity and pressure distribution in a solar chimney can be affected by var-

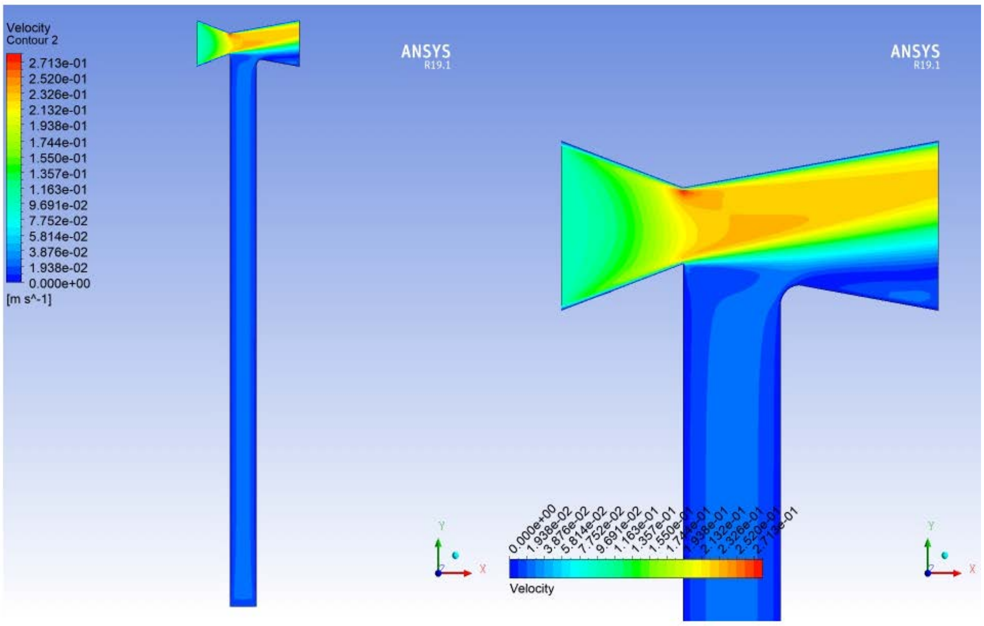


Fig. 7. Velocity contours in the CFD domain ($V_{\text{wind}} = 0.1$ m/s).

ious factors, such as the height and other dimensions of the chimney, the solar irradiation, the surrounding environment, and weather conditions. In this simulation, we consider only the effect of wind by varying its velocity and set a constant heat flux at the absorber.

In this case, especially under certain assumptions and simplifications, the relationship between wind speed and ventilation flow rate is practically linear as shown in Fig. 8. This means that if the wind speed doubles, the ventilation flow rate would also double, assuming other factors remain constant. We note also that with no wind (wind velocity $= 0$ m/s) the ventilation flow rate is 0.0012 m^3/h.

5 CONCLUSION

Convergent-Divergent (CD) geometry with a vane tail at the chimney discharge area is proposed to create a pressure drop and harness the wind's advantages for any direction and speed; a numerical simulation using Ansys Fluent code is achieved to predict the wind impact on solar chimney (SC) performances. The main results are:

- The wind has a significant effect on the chimney efficiency.

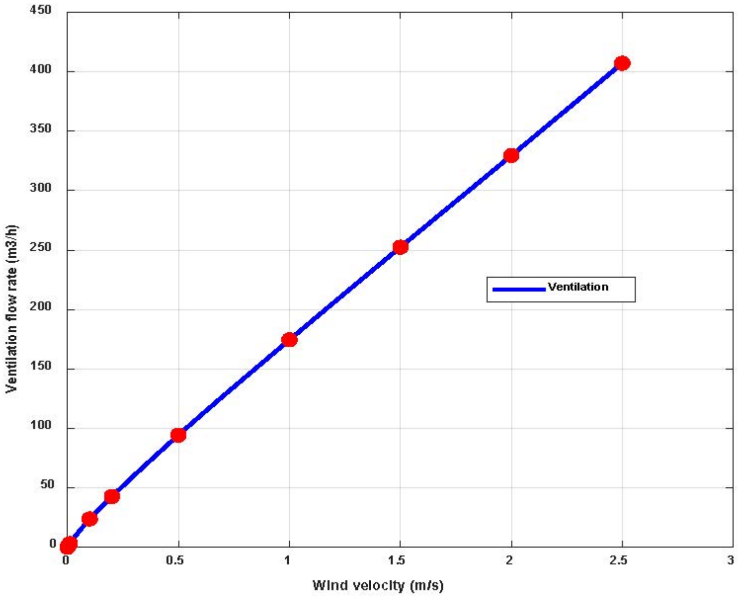


Fig. 8. Ventilation flow rate variation with wind velocity.

- With this design, the chimney can provide the ventilation without the solar assistance mainly for low or no insolation periods as well as at night.

- Linear relationship between wind speed and ventilation flow rate can serve as: the ventilation flow rate doubles if the wind speed doubles.

REFERENCES

[1] C. HOY-YEN, B.R. SAFFA, Z. JIE (2010) Review of passive solar heating and cooling technologies. *Renewable and Sustainable Energy Reviews* **14** 781-789.

[2] F. MANZANO-AGUGLIARO, F.G. MONTOYA, A. SABIO-ORTEGA, A. GARCÍA CRUZ (2015) Review of bioclimatic architecture strategies for achieving thermal comfort. *Renewable and Sustainable Energy Reviews* **49** (9) 736-755.

[3] K. RAKESH, L. CHENGWANG (2011) Solar chimney – A passive strategy for natural ventilation. *Energy and Buildings* **43** 1811-1819.

[4] C. AFONSO, A. OLIVEIRA (2000) Solar chimneys: simulation and experiment. *Energy and Buildings* **32** 71-79.

[5] C. JIMÉNEZ-XAMÁN, J. XAMÁNÁ, M. GIJON-RIVERA, I. ZAVALA-GUILLÉN, F. NOH-PAT, E. SIMÁ (2020) Assessing the thermal performance of a rooftop solar chimney attached to a single room. *Journal of Building Engineering* **31** 101380.

[6] Z. Xinyu, J. Zhang, Q. Menghao (2017) Experimental and Numerical Studies of Solar Chimney for Ventilation in Low Energy Buildings. *Procedia Engineering* **205** 1612-1619.

[7] E.A. Karima, W.M. Saif (2012) Experimental and numerical studies of solar chimney for natural ventilation in Iraq. *Energy and Buildings* **47** 450-457.

[8] G. Gan, S.B. Riffat (1998) A numerical study of solar chimney for natural ventilation of buildings with heat recovery. *Applied Thermal Engineering* **18** 1171-1187.

[9] H. Yicun, L. Huang, L. Angui (2019) Experimental and theoretical study of solar chimneys in buildings with uniform wall heat flux. *Solar Energy* **193** (2019) 244-252.

[10] F.M. Boudjema, H. Benzenine (2023) Impact of variable passage section and zigzag absorber on the thermal performance of a thermo-solar converter for drying. *Journal of Theoretical and Applied Mechanics* **53**(1) 66-81.

[11] K. Jing, N. Jianlei, L. Chengwang (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.

[12] E.P. Sakonidou, T.D. Karapantsios, A.I. Balouktsis, D. Chassapis (2008) Modeling of the optimum tilt of a solar chimney for maximum air flow. *Solar Energy* **82** 80-94.

[13] T.V. Nguyen, Y. Quoc Nguyen (2021) Performance of a solar chimney configuration to achieve equal flow rate for ventilation of three-story building. *IOP Conf. Series: Materials Science and Engineering* **1109** 012001.

[14] S. Long, Z. Guomin, Y. Wei, H. Dongmei, C. Xudong, S. Sujeeva (2018) Determining the influencing factors on the performance of solar chimney in buildings. *Renewable and Sustainable Energy Reviews* **88** 223-238.

[15] N. Gao, Y. Yan, R. Sun, Y. Lei (2022) Natural Ventilation Enhancement of a Roof Solar Chimney with Wind-Induced Channel. *Energies* **15** 6492.

[16] (accessed December 2022) https://www.carboun.com/sustainable-design/a-damascus-school-revives-traditionalcooling-techniques.

[17] M. Eghtedari, A. Mahravan (2021) evaluation a hybrid passive cooling system for a building using experimental and commercial software (design builder). *Journal of Renewable Energy and Environment* **8**(2) 74-80.

[18] L.C. Haw, O. Saadatian, M.Y. Sulaiman, S. Mat, K. Sopian (2012) Empirical study of a windinduced natural ventilation tower under hot and humid climatic conditions. *Energy and Buildings* **52** 28-38.

[19] O. Belhadj, H.Benzenine, R. Saim (2021) Experimental Analysis of Height to Base Length Effect in Trapezoidal Prism Inclined Solar Chimney. *Periodica Polytechnica Mechanical Engineering* **65**(1) 47-55.