Global Journal of Research in Engineering & Computer Sciences
ISSN: 2583-2727 (Online)
Volume 05| Issue 02 | March-April | 2025
Journal homepage: https://gjrpublication.com/gjrecs/
Research Article

CFD analysis of the performance of a wind catcher system coupled with solar chimney

*Iman Shemshadi

Ms.c, Department of Mechanical Engineering, Payame Noor University, Tehran, Iran

DOI: 10.5281/zenodo.15213772 **Submission Date:** 10 March 2025 | **Published Date:** 15 April 2025

Corresponding author:** *Iman Shemshadi
Ms.c, Department of Mechanical Engineering, Payame Noor University, Tehran, Iran
ORCID :0000-0002-8664-4602

Abstract
*The wind deflector is one of the indigenous elements that is widely used in hot, hot, dry and hot humid climates, especially in the Middle East countries. A system consisting of a wind deflector and a solar chimney has been designed. The purpose of this simulation with CFD is to compare the performance of the wind deflector in the presence of a solar chimney. The simulation is carried out in the range of 0° for a uniform wind velocity of (0.6 m/s -1.5 m/s) the entrance boundary condition is inlet velocity and the exit boundary condition is outlet pressure. The dimensions of the room connected to the system are 1.5 * 1.25 * 1m (length, width and Height). This room is a closed room with no opening. The lower part of the chimney (back to the wind) is blocked. The absorber plate is a dark aluminum plate with a light absorption coefficient of 90%. The rest of the walls of the system are isolated and the system is adiabatic. Due to the effect of the buoyancy force in the natural movement of air in the wind-solar chimney, it has a greater effect in reducing the room temperature. Also, the decrease in humidity occurred at low air speeds in a tangible way, and the buoyancy force caused by the solar chimney is more obvious.*

Keywords: *wind catcher, solar chimney, CFD, simulation.*

INTRODUCTION
The idea of a Wind catcher originated Two thousand years ago in the Middle East [1] and referred to as wind tower. Domed and vaulted ceilings were widely used by builders and architects throughout the Middle East and other hot arid regions such as Dubai, Jordan, Bahrain, Oman, Iran and Pakistan in traditional and vernacular buildings [2 Today, wind catcher is still seen in various designs, including one-way, two-way, and multi-directional in many countries. As can be seen in Figure 1, such roofs, which are made of brick with a plaster coating, usually have a small opening near the top of the gable walls of the arches, which causes ventilation and removes hot air from a higher height. The shape, height and internal structure of wind towers, especially in desert areas, in addition to providing cooling load, is the crystallization of Iranian culture and architectural potential [3, 4].

Fig.1. Persian wind catcher

The fluid flow of air on the roof causes a significant pressure difference around the roof, which induces the air flow to the interior of residential areas. The air enters from the entrance of the wind tower with a positive pressure coefficient and exits from the external paths with a negative pressure coefficient [5]. It is possible that in a certain area, the wind blows more in a certain direction, so wind towers usually have several traps to transfer part of the wind to the interior space [6, 7]. Elmualim and Hazim [8] have done simulation and laboratory studies on the performance of square wind tower. As a result of these studies, they explained that the sharp edges of the square cause a large area of separation and stagnation of flow and a higher pressure difference throughout the device. Montazeri and Dehghan [9] analyzed the flow in wind towers using two-dimensional hydrodynamic computational fluid dynamics. They showed that the separated flow near the bottom edge of the wind tower aperture significantly reduces the inductive capacity of the wind tower. Elmualim [10] analyzed the performance of an air tower experimentally and showed that, in general, the amount of ventilation in the selected design (closed window room) is more than in the open window room. Montazeri et al. [11] studied a one-way wind tower in a simulation and experimental way. They evaluated the amount of induced flow to the wind tower (or its external flow) and the pressure coefficient on different surfaces. His results showed that one-way wind towers perform well in full wind direction .Ghadiri et al. [12] after extensive numerical research on the implementation of square wind towers with different dimensions in hot and dry regions, proposed the use of wind towers as an option for a green ventilation system with the ability to increase indoor air quality with minimal energy consumption In another research, Ghadiri et al. [13] investigated the effect of the geometry of traditional wind towers on the internal heating behavior of the building. By simulating square wind towers with different values, they presented a suitable model about how traditional wind towers work.

The present study is related to heat transfer and numerical simulation of air flow in solar chimneys. The innovation of the current project includes a combination of wind tower and solar chimney.

Research approach

Accurate information about a physical process is often obtained by practical measurement. The laboratory research conducted on a device whose dimensions are the same as the original device is used to predict how similar versions of the mentioned device will work under the same conditions, but in most cases such tests are not performed due to the large size. The devices are very expensive and often impossible, so the tests are performed on models with sizes on a smaller scale, although here, too, the problem of expanding the information obtained from the smaller sample is always simulating all aspects of the original device. they do not, and often important aspects of the model tests are omitted, these limitations further reduce the usefulness of the test results, finally, it should be remembered that in many cases, there are serious measurement problems. And measuring instruments are not free from errors. A theoretical prediction will make the maximum use of the results of the mathematical model and in comparison, with it, it will use the experimental results less. For the physical processes in question, basically, the mathematical model consists of a series of differential equations. If classical mathematics methods were to be used in solving these equations, it would not be possible to predict many useful phenomena. With a little attention, it is clear that only for a small number of scientific problems, the necessary equations can be found with the number of unknowns. In addition, their answers often include infinite series, special functions, non-algebraic equations, eigenvalues, etc. As it is possible, their numerical solution is not an easy task. Fortunately, the development of numerical methods and the availability of large processors have ensured that the concepts of a mathematical model can be used for almost any practical problem.

Computational Fluid Dynamics

Numerical solution of fluid flow equations is one of the most important issues in engineering. Mechanical engineers are highly dependent on experimental tests in the design of heat exchangers, oil and gas pipelines, refineries, etc. Civil engineers, in turn, depend on testing their models in hydraulic laboratories to build hydraulic structures. To design chemical reactors, chemical engineers need to understand fluid flow and perform experiments in fluid environments with chemical reactions. What has been mentioned shows that basically, the design and construction of all devices related to fluid flow requires fluid tests related to that device. Experiments have been widely used for several decades to design and manufacture various devices involved in fluid flow. Computational fluid dynamics is the analysis of systems including fluid flow, heat transfer and accompanying phenomena, such as chemical reactions, based on computer simulation. CFD is a very powerful method, so that it includes a wide range of industrial and non-industrial applications. In order to use CFD methods, it is necessary to discretize the flow field using a grid, and to simulate the flow field, the computational domain must be divided into small elements so that the aforementioned equations can be solved on them. Grid generation is considered one of the most important parts of numerical solution, which is very influential in the accuracy of numerical solution, and more than 50% of the analysis time of a CFD problem is devoted to its grid generation. Today, CFD methods are considered a powerful tool in the design process, along with manufacturing laboratories, and since the cost of numerical methods is very low compared to experimental tests, in many cases, numerical methods are an alternative to conventional methods. have also been experimental. Today, computational fluid dynamics is so developed that it is capable of solving infrasonic, transsonic, ultrasonic, supersonic, reaction flows, multiphase flows, etc. Nowadays, due to the ever-increasing expansion of computing power of computers, it has become possible to solve flow equations for complex geometries that include real and industrial cases.

GOVERNING EQUATION

The equations governing the fluid flow or the Navier-Stokes equations, which correctly predict the flow phenomena in the fluid, were independently obtained by Navier and Stokes. Of course, before obtaining the mentioned equations, scientists such as Archimedes, Newton and Euler had worked in the field of fluid flow and found results in this field, but the work of Navier and Stokes in finding the complete equations governing the flow is outstanding and significant. Navier-Stokes equations are obtained by using the principle of conservation of mass, Newton's second law and the principle of conservation of energy and give a complete description of fluid movement. These equations are able to simulate multiphase flows, flows with chemical reaction and flows at very high speeds. The aforementioned equations are a set of nonlinear partial differential equations that are completely dependent on each other and have a very complex nature. The non-linear nature of the equations makes it impossible to solve the mentioned equations with analytical methods and it is difficult to solve them with numerical methods, especially when dealing with turbulent flows, this issue becomes more important. Most of the real engineering flows are turbulent, so solving turbulent flows in engineering is of special importance. In general, Navier-Stokes equations is a complete system of equations that is capable of simulating all types of flow, these equations are also capable of simulating turbulent flows, but for the full simulation of turbulent flow, a very fine grid is needed, which is beyond the capabilities of today's computers. Is. Therefore, in order to solve the turbulent engineering flows, other methods are used, among them, simulation using the averaged Reynolds equations or simulation of large eddies. Numerical solution of fluid flow equations at high Reynolds numbers, which is naturally the flow at these numbers is also turbulent, is still one of the problems that researchers and engineers struggle with, and it seems that this problem will continue to bother researchers and engineers for a long time. busy himself the most important fact governing turbulent flows is that said flows are the most complex type of fluid flow. The differential equations that govern a flow of fluids based on the Navier-Stokes equation:

Continuity equation:

$$\frac{\partial \rho}{\partial \mathbf{t}}+\frac{\partial}{\partial \mathbf{x}_i}\left(\rho \mathbf{u}_i\right)=\mathbf{0} \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial \mathbf{t}}\left(\rho \mathbf{u}_i\right)+\frac{\partial}{\partial \mathbf{x}_i}\left(\rho \mathbf{u}_i \mathbf{u}_i\right)=-\frac{\partial \rho}{\partial \mathbf{x}_i}+\frac{\partial}{\partial \mathbf{x}_i}\left[\mu\left(\frac{\partial \mathbf{u}_i}{\partial \mathbf{x}_i}+\frac{\partial \mathbf{u}_j}{\partial \mathbf{x}_j}-\frac{\mathbf{2}}{\mathbf{3}}\delta_{ij}\frac{\partial \mathbf{u}_k}{\partial \mathbf{x}_k}\right)\right]+\frac{\partial}{\partial \mathbf{x}_j}\left(-\rho \overline{\mathbf{u}'_i \mathbf{u}'_j}\right) \tag{2}$$

Energy equation (3)

$$\frac{\partial}{\partial x_i}\left[u_i \rho\left(h+\frac{1}{2}u_j u_j\right)\right]=\frac{\partial}{\partial x_j}\left[k_{eff}\frac{\partial T}{\partial x_j}+u_i(\tau_{ij})_{eff}\right],$$

$$k_{eff}=K+\frac{c_p \mu_t}{Pr_t}$$

Turbulence in laminar flows first occurs in response to fluid instabilities at high Reynolds numbers. In other words, the turbulent flow alone cannot be stable, but must receive energy from its environment. One of the main sources of turbulent flow energy is shear stress, another source can be buoyancy. If no energy is supplied to an initially turbulent flow, the flow will automatically calm down. High penetration rate is one of the characteristics of turbulent flows. In fact, in turbulent flows, the rate of transfer and mixing of momentum, kinetic energy and other quantities such as heat and other scalars is much higher than in smooth flows. In the study of the Navier-Stokes equations governing the flow of liquids or gases, which can be a good description of turbulent flows, it is shown that the smallest length scales in turbulent flows are much larger than the mean free distance between molecules, and therefore the turbulent flow is a continuous medium, the main mechanism The creation and stability of turbulence is the mutual effect of velocity gradient and vorticity. Vorticity is the curl of the velocity field, and in fact, the turbulent flow can be understood according to the vorticity transfer equation. As mentioned, Navier-Stokes equations are a system of nonlinear equations. Solving these equations with analytical methods is currently not possible, so numerical methods have been developed to solve these equations. As mentioned, most of the flows that we want to study are turbulent. But the numerical solution of turbulent flows faces many problems. The most important problem in solving turbulent flow problems is that these flows are highly dependent on the Reynolds number. Reynolds number, which is defined as the ratio of viscous forces to inertial forces, is an important factor in determining the turbulent flow. As mentioned, one of the characteristics of turbulent flows is that turbulent flow occurs at high Reynolds numbers. In fact, fluid flows are disturbed at a certain Reynolds number depending on external disturbances and the roughness of solid surfaces, which is called the critical Reynolds number. The mentioned mechanism is very complicated and in fact the flow goes through a transient state and then becomes disturbed. As it was said, turbulent flows are dependent on the Reynolds number, which means that the characteristics of the flow change with the change of the Reynolds number. There is a wide range of length and time scales in the turbulent flow, small length scales have less energy and large length scales contain more energy. In the turbulent flow of small scales, the flow is completely dependent on the Reynolds number of the flow, but the larger scales of the flow are more affected by the flow geometry and their dimensions are within the limits of the flow geometry scales. In turbulent flows, energy is transferred from large flow eddies to smaller flow eddies through the energy cascade mechanism and is wasted through the viscous dissipation mechanism. The Reynolds number of the flow determines the smallest length scales of the flow, therefore, to fully solve the fluid flow equations, the mesh size of the Use the smallest size of the current scale. On the other hand, from turbulent flow theories, we know that with the increase of the Reynolds number, the size of the smallest flow scales decreases and the need to refine the grid in order to fully solve the flow field and capture all the flow scales increases and is able to fully solve the fluid flow due to the limitations of the computing power of computers. Today, we are not complicated in practical matters. Therefore, there is a need for methods that can be used to effectively and usefully simulate the turbulent flow. Simulation of turbulent fluid flow using computer is a relatively new field in engineering sciences and still has a long way to develop and improve turbulent flow solving methods. A confusing model that can be used for all situations and problems of the offender. Unfortunately, it does not exist, and the choice of the confusing model depends on considerations such as flow physics, simulation experiences for specific problems, the required accuracy, the power of computing resources (computer power) and the time available to perform calculations. Of course, to choose a suitable confusion model, you must know the capabilities and limitations of all models and states. In complex geometries, the Navier-Stokes equations cannot be completely solved, but to solve the Navier-Stokes equations So that the turbulent flow fluctuations are not directly included in the equations, there are generally two methods: a) Reynolds averaging method (time averaging) RANS. b) The method of simulating large waves using LES filtering. In this project, Reynolds averaging method is used.

In turbulent flow conditions the instantaneous governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove small scales, resulting in a modified set of equations that are computationally less expensive to solve. has less The CFD solver therefore provides several turbulent models to solve specific application problems, as shown below. It is a semiempirical model and the derivation of the model equations relies on phenomenological considerations and empiricism. The standard K- e Model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (e). The model transport equation for k is derived from the exact equation, while the model transport equation for e was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the K- e Model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard K- e Model is therefore valid only for fully turbulent flows. The turbulence kinetic energy, k and its rate of dissipation, e are obtained from the following transport equations.

At high speeds the turbulence viscosity is according to Equation (4), which is the same equation that the standard $k-\varepsilon$ model uses to calculate μt.

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{4}$$

The RNG model uses the following transfer equations for ε and k:

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}\right] + 2\mu_t E_{ij}.E_{ij} - \rho\varepsilon - Y_M \tag{5}$$

$$\frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}\right] + C^*_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij}.E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon \tag{6}$$

The values and constants used in the relation are as follows:

$$\mu_{eff} = \mu + \mu_t \tag{7}$$

$$C^*_{1\varepsilon} = C_{1\varepsilon} - \frac{\eta(1-\eta/\eta_0)}{1+\beta\eta^3} \tag{8}$$

$$\eta = (2E_{ij}.E_{ij})^{1/2}\frac{k}{\varepsilon} \quad (9)$$

$$E_{ij} = \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \quad (10)$$

$$Y_M = 2\rho\varepsilon M_t^2, M_t = \sqrt{\frac{k}{\gamma RT}} \quad (11)$$

$$R_\varepsilon = \frac{C_\mu \rho \eta^3 (1-\eta/\eta_0)}{1+\beta\eta^3}\frac{\varepsilon^2}{k} \quad (12)$$

$$\eta_0 = 4.377, \beta = 0.012$$

$$C_\mu = 0.0845, \alpha_k = \alpha_\varepsilon = 1.39, C_{1\varepsilon} = 1.42, C_{2\varepsilon} = 1.68, Pr_t = 0.85$$

Eiji, deformation rate tensor, YM represents the share of oscillating expansion in incompressible turbulent flow. This term becomes important in cases where the Mach number is high and is automatically removed from the equations for low Mach flows. αk and ε are inversely related to the parenthesis number of equations k and., Respectively. Mt is used as the Mach number of the compressible turbulent flow, γ is the ratio of the specific heat coefficient of constant pressure to constant volume and $\eta 0$, $C\mu$, $C1\varepsilon$, $C2\varepsilon$, η, $C\upsilon$ are the constants of the turbulent flow model $k\text{–}\varepsilon$. The RNG $k\text{–}\varepsilon$ model has an additional term in the equations compared to the Standard $k\text{–}\varepsilon$ model, which sufficiently increases the computational accuracy at high speeds. It has good effects on tangential and rotational currents and increases the accuracy of calculations in rotational currents. The RNG $k\text{–}\varepsilon$ model provides an analytical formula for calculating the turbulent flow Prandtl number, while the standard model uses constant values. The Standard $k\text{–}\varepsilon$ model provides suitable solutions only for flows with very high Reynolds numbers, while the RNG $k\text{–}\varepsilon$ model provides relatively good solutions for low Reynolds equations by providing a differential equation to calculate the effective viscosity. This causes the accuracy of the calculations to be sufficiently high in the area near the wall. While using the standard model in the area near the wall is not appropriate.

THE WIND CATCHER SIMULATION

To simulate and analyze a four-sided wind tower, a geometry is numerically generated from the wind tunnel test. The simulation is carried out in the range of 0° for a uniform wind velocity of (0.6 m/s -1.5 m/s) . According to Fig. 2, the entrance boundary condition is inlet velocity and the exit boundary condition is outlet pressure. The dimensions of the room connected to the system are 1.5 * 1.25 * 1m (length, width and Height). This room is a closed room with no opening. The lower part of the chimney (back to the wind) is blocked. So that the air does not enter the chimney duct directly from the wind deflector and leaves the room through the air inlet valve after entering the room. The absorber plate is a dark aluminum plate with a light absorption coefficient of 90%. The rest of the walls of the system are isolated and the system is adiabatic. It is assumed that the walls of the room do not participate in heat transfer, so the solar chimney or the back side to the windshield, is solely responsible for the effects of heat buoyancy on the air.

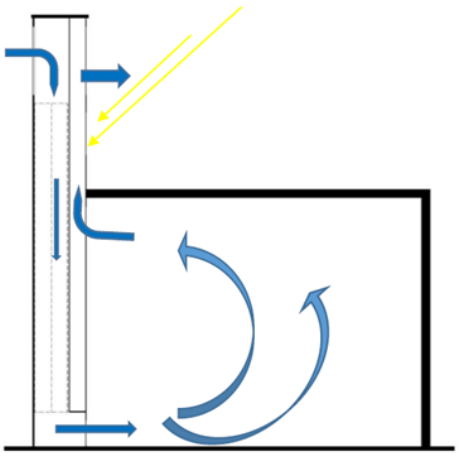

Fig.2. Schematic form of computational domain

TABLE I. Geometrical specifications of the system

	The geometry	Dimensions
1	Windshield inlet valve	0.1*0.4 =0.04 m^2
2	Chimney air outlet to the environment	0.1*0.4 =0.04 m^2
3	Windshield vent	0.16*0.15 =0.024 m^2
4	Solar absorber plate length	0.25 m
5	Solar absorber plate width	0.1 m
6	Room air inlet to the chimney	0.16*0.15=0.024 m^2
7	height	2.5 m

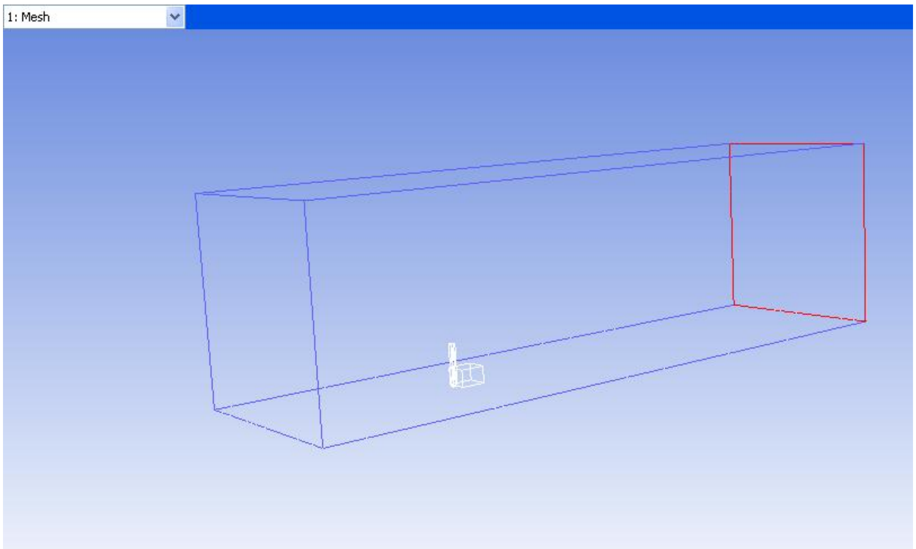

Fig.3. computational domain

GRID GENERATION

Due to the complexity of the geometry discussed in this research, pyramidal cells have been used. According to Figure 4, in this research, unorganized pyramidal elements were used in the production of the network. The network was finer near the places where the gradient of temperature, pressure and speed is more intense, so that these changes can be simulated more accurately.

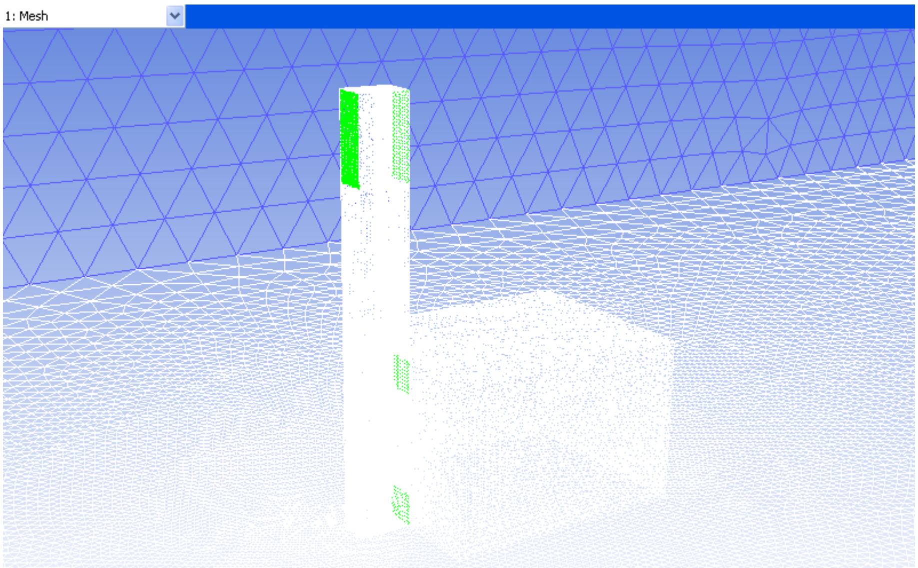

Fig.4. Computational domain grid generation

Fig.5. shows the velocity distribution at the bottom and the top of channel of the wind catcher. The air motion within wind catcher and room is also visible. the resulting vortex behind the wind tower indicates a negative pressure zone, which leads to the suction of induced air from other traps after passing through the lower space. An increased flow velocity is observed in wind catcher due to reduction in cross sectional area.

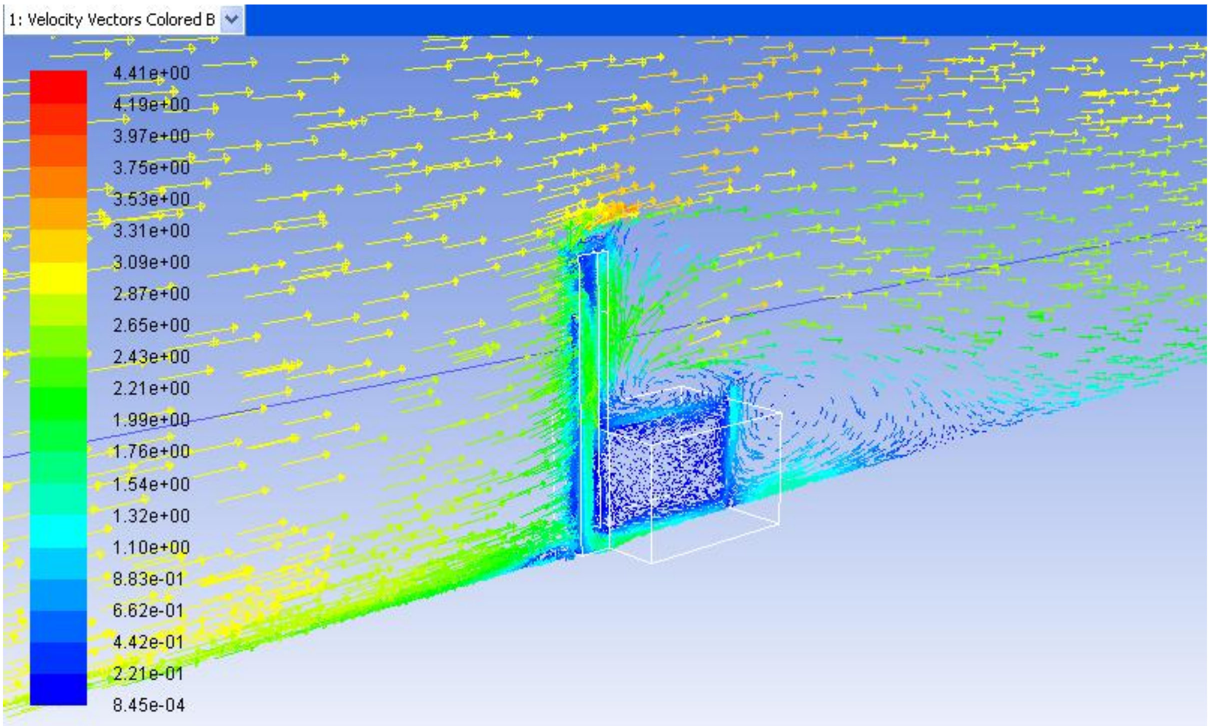

Fig.5. Speed vector contour in parallel cross section in wind speed 1.5 m / s

Fig. 6 shows the flow separation and pressure reduction in front of wind catcher, which is the main reason for air suction into the wind catcher. The pressure contours signify gentle pressure drops in regions of higher flow velocities on the back surface of wind catcher, whereas relatively higher pressure is observed within the room due to the slowing down of air flow.

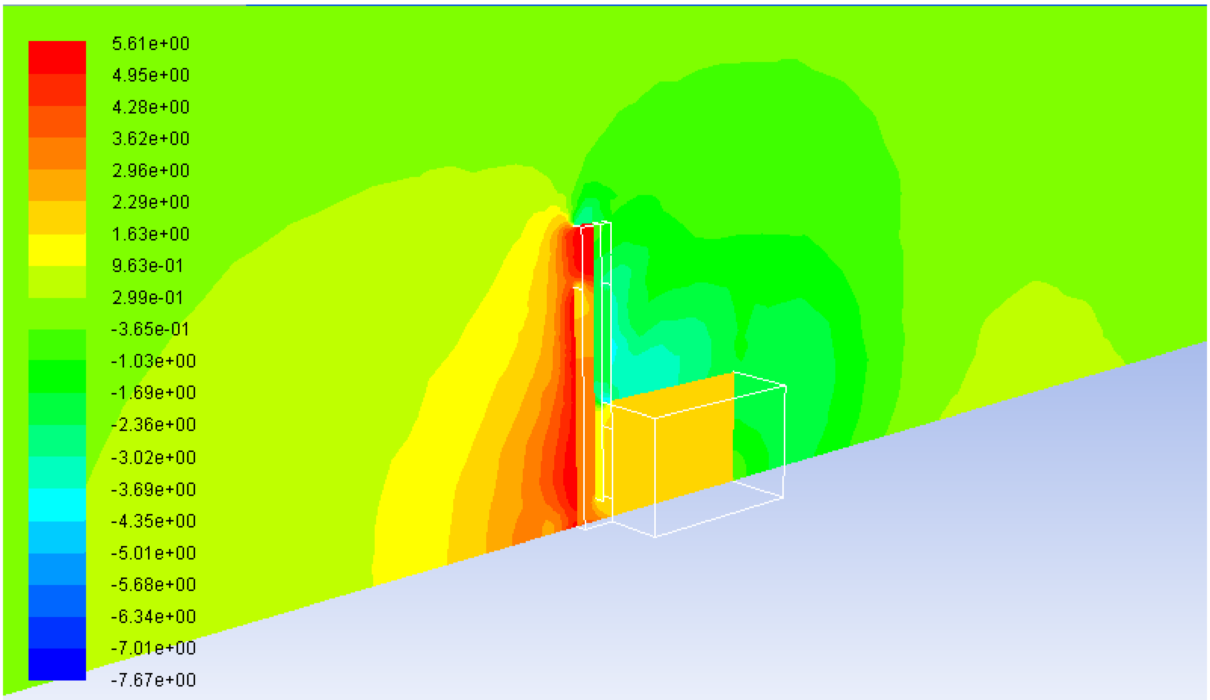

Fig.6. static pressure in wind speed 1.5 m / s

According to Fig. 7, The results of temperature in terms of air velocity indicate the temperature reduction in the room. Reducing temperature with increasing speed from 1.2 m/s to 1.5 m/s is mild by comparing wind catcher-solar chimney with simple wind catcher, it can be concluded that temperature has decreased due to the use of solar chimney. This decrease in temperature is higher due to the effect of solar chimney at low wind speed. Because of the effect of Buoyancy force in the natural convection of the air in the wind catcher-solar chimney, it has a greater effect on the reduction of room temperature.

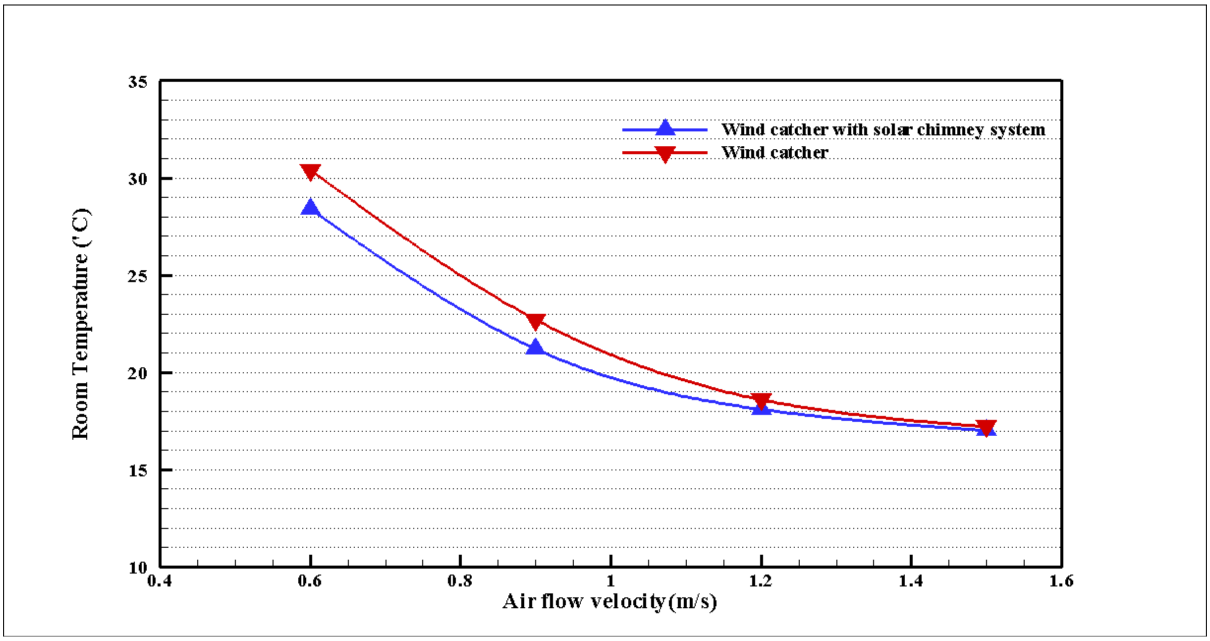

Fig.7. Changes in room temperature in terms of wind speed

Fig. 8. indicates changes in humidity in terms of air velocity. With increasing air speed, the amount of humidity inside the room decreases. By comparing the wind catcher-solar chimney with simple wind catcher, it can be concluded that the humidity inside the room has decreased due to the use of solar chimney. As we expected, due to the impact of the natural convection of the air in the wind catcher-solar chimney, it has a greater effect of reducing the air humidity of the room. This reduction in humidity due to the use of solar chimney effects has been reduced by about 10%. This highest decrease in low air speeds occurred, which is more pronounced Buoyancy force.

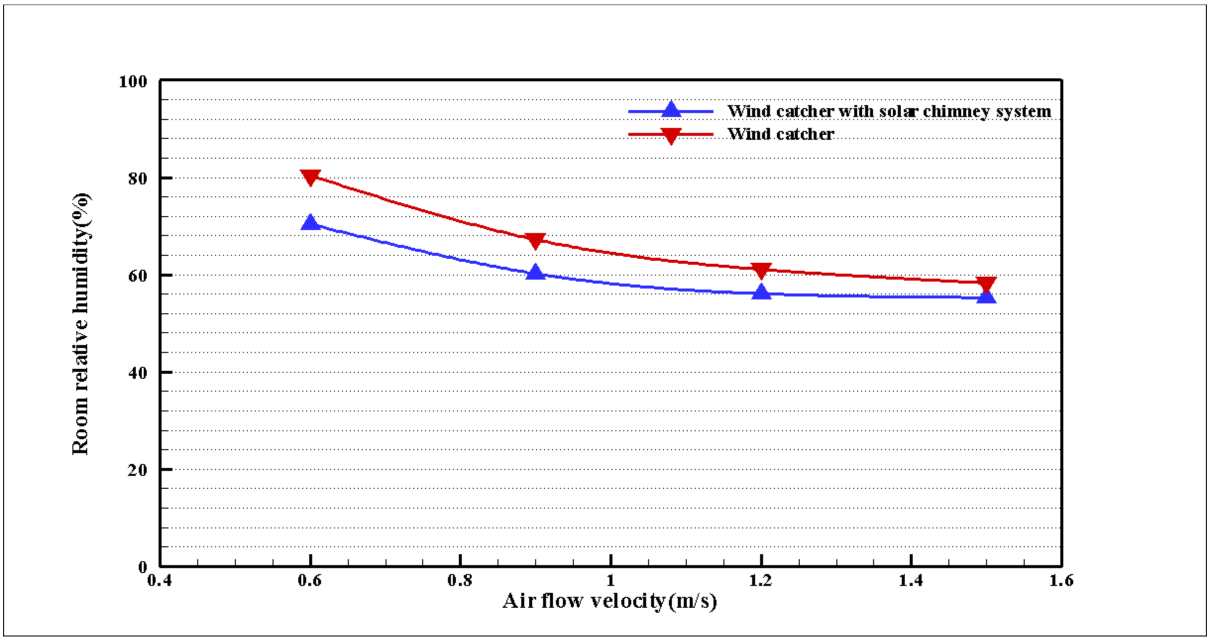

Fig.8. Changes in room humidity in terms of wind speed

Conclusion:

In this simulation, the wind deflector performance was investigated in the presence of a solar chimney. Ansys Fluent software was used for numerical simulation. Continuity, momentum and turbulence equations were solved. It was shown that due to the effect of buoyancy on the natural movement of air in the wind-sun chimney, it had a greater effect on reducing the room temperature. Also, the decrease in humidity at low air speeds has been noticeable and the buoyancy force caused by the solar chimney has been more evident. By comparing the solar winder-chimney with the simple winder, it was concluded that the humidity in the room has decreased due to the use of the solar chimney. Due to the effect of natural air convection in the wind-solar chimney, it has had a greater effect on reducing the humidity in the room.

References:

1. Bahadori, Mehdi N. (August 1994). "Viability of wind towers in achieving summer comfort in the hot arid regions of the Middle East". Renewable Energy.
2. A. A'zami. Badgir in traditional Iranian architecture- Retrieved on 17-07- 2007.
3. Fathy H., Natural Energy and Vernacular Architecture: Principles and Examples with Reference to Hot Arid Climates (1986) ISBN-10: 0226239179, The University of Chicago Press.
4. Gage S.A., Graham J.M.R., Static Split Duct Roof Ventilation, Building Research & Information (2010) 28(4): 234–44.
5. Bahadori M.N., Viability of Wind Towers in Achieving Summer Comfort in the Hot Arid Regions of the Middle East, Renewable Energy (1994) 5(5–8): 879–892.
6. Bansal N. K., Mathur R., Bhandari M. S., A Study of Solar Chimney Assisted Wind Tower System for Natural Ventilation in Buildings, Building and Environment (1994) 29(4): 495–500.
7. Liu L., Mak C.M., The Assessment of the Performance of a Windcatcher System Using Computational Fluid Dynamics, Building and Environment (2007) 42(3): 1135–1141.
8. Elmualim A.A., Hazim B.A., Wind Tunnel and CFD Investigation of the Performance of Windcatcher Ventilation Systems, International Journal of Ventilation (2002) 1(1): 53–64.
9. Montazeri H, Dehghan A.A., Numerical Investigation of Induced Air Inside a Typical Building Through a Wind Catcher as a Passive Cooling Component, 13th Fluid Dynamics Conferences Yazd Iran (2006).
10. Elmualim A.A., Verification of Design Calculations of a Wind Catcher/Tower Natural Ventilation System with Performance Testing in a Real Building, International Journal of Ventilation (2006) 4(4): 393–404.
11. Montazeri H., Montazeri F., Azizian R., Mostafavi S., Two-Sided Wind Catcher Performance Evaluation Using Experimental, Numerical and Analytical Modeling, Renewable Energy (2010) 35(7): 1424–1435.
12. M.H. Ghadiri, N. L. N. Ibrahim, M. F. Mohamed, ―Performance evaluation of four-sided square windcatchers with different geometries by numerical method‖, Engineering Journal, vol. 17, issue 4, pp. 9-18, 2010.
13. M. H. Ghadiri, N. L. N. Ibrahim, R. Aayani, ―The effect of windcatcher geometry on the indoor thermal behaviour‖, Australian Journal of Basic and Applied Sciences, vol. 5, issue 9, pp. 381-385, 2010.

CITATION

Iman S. (2025). CFD analysis of the performance of a wind catcher system coupled with solar chimney. In Global Journal of Research in Engineering & Computer Sciences (Vol. 5, Number 2, pp. 63–71).
https://doi.org/10.5281/zenodo.15213772