International Journal of Science and Research Archive

eISSN: 2582-8185

Cross Ref DOI: 10.30574/ijsra

Journal homepage: https://ijsra.net/

(Review Article)

Numerical study of flow characteristics in solar air collectors with turbulators

Umidjon Ibragimov [1], Gulam Uzakov [1] and Jobir Kodirov [2,*]

[1] Karshi State Technical University, 180019, Khonobod st., 19, Karshi, Uzbekistan.
[2] Department of Heliophysics, Renewable Energy Sources and Electronics, Bukhara State University, 200117, M. Ikbol st., Bukhara, Uzbekistan.

International Journal of Science and Research Archive, 2025, 15(01), 1325-1330

Publication history: Received on 15 March 2025; revised on 22 April 2025; accepted on 24 April 2025

Article DOI: https://doi.org/10.30574/ijsra.2025.15.1.1203

Abstract

This article presents the results of the study of flow characteristics, flow rate and pressure changes from the air duct when turbulizers are used to accelerate heat-hydrodynamic processes in solar air collectors. The results of this study were obtained using the software tool COMSOL Multiphysics 6.1, which is recommended to be used in the numerical research and design of turbulizer solar air collectors.

Keywords: Solar energy; Solar air collector; Turbulizer; Air flow; Air velocity; Pressure

1. Introduction

Solar air collectors (SAC) are widely used in heating buildings [1-3], drying agricultural products [4-7] and other industries, as they are simple in structure, convenient in installation and easy in service. However, due to the low heat transfer between the absorber plate and the airflow and the high heat loss to the environment, the efficiency of SAC is much lower. Currently, a large number of theoretical studies have been carried out to improve the thermal efficiency and productivity of SACs. Researchers have proposed increasing the rate of heat transfer by installing various modifications between the absorber plate and the transparent coating of the SAC. Among the various modifications, the installation of barrier-shaped turbulizers in the transverse position in the direction of air flow is based on the highest efficiency [9].

The research work of Yeh et al [10] experimentally investigated the effect of the geometric dimensions of the turbulizer on the characteristic of Sac. According to the results, the productivity of Sac increased with an increase in the length of the turbulizer. Moummi et al [11] had developed a sac whose turbulizer was fastened perpendicular to the airflow to produce high turbulence of the flow. According to the results, the thermal characteristic of SAC has significantly improved. Bensaci et al [12] conducted an experimental study of the effects of transversely located turbulizers on the heat transfer coefficient and the FIK value of Sac. According to the results, SAC with turbulizers installed in all parts of the air duct had the highest productivity. Romdhane [13] had proposed a new sac with turbulizers installed in the air duct. He compared the different positioning schemes of turbulizers in terms of Sac efficiency. However, data on flow characteristic and heat transfer in this Sac are not cited in the research work. Also, in the research work, it was noted that when turbulizers are placed in a chess order and in the correct order, there is a significant increase in the thermal efficiency of Sac.

In order to improve the thermal efficiency of the SAC, it has been proposed to drill holes in the turbulator without altering the macrosation, and to install cross-pyramid-shaped soplos with a suitable size for the hole shaft. This approach is based on the principle of wind protection walls, such that the cut pyramid-shaped soplos will not leave large piles behind the turbulator. Reactive currents form when the air flow passes through a cross-pyramidal sopro,

* Corresponding author: Jobir Kodirov

accelerating turbulence and interference at the outlet of the channel, thereby eliminating local shocks after the turbulator. This effect eliminates hotspots, increases the mixing intensity of air flows in the main channel with the air flowing out of the sopro, and reduces heat loss. Additionally, reactive currents enhance convective heat transfer over the air flow and the absorber plate.. As a result, the amount of heat in the air increases and Sac's high efficiency is achieved.

In this article, we conducted a comprehensive study on the effect of various parameters, such as turbulizers, holes, and confusors, on the hydrodynamic process in a Sac and its flow characteristics. We used numerical modeling software to simulate the behavior of the system and investigate the relationships between the different variables. Six numerical models were created, including two-dimensional and three-dimensional simulations. A statistical analysis was then performed to examine the effects of different turbulizer shapes, numbers, and airflow velocities on the Sac's hydraulic process and flow characteristics, including flow patterns, Reynolds numbers, velocities, turbulent kinetic energy, turbulent viscosity, and pressure losses. The results of our study provide valuable insights into the mechanisms and theoretical foundations for changing the air flow friction coefficient in the duct, which is essential for determining the thermal and hydraulic efficiency of a turbulizer Sac.

2. Material and methods

- **Theoretical research methodology:** Theoretical studies aimed at improving hydrodynamic processes and flow characteristics in SAC, which involve the installation of turbulizers with various geometric shapes and parameters, are based on the use of the Comsol Multiphysics 6.1 software tool [14]. The numerical modeling used in this dissertation study is based on the following assumptions: the fluid flow and heat exchange are three-dimensional, the flow is stable and incompressible, heat loss through the back and side walls of the SAC body is negligible, heat loss at the entry and exit of SAC is also negligible, the thermal-physical properties of liquid (air) and solid (absorber plate and turbulizer) remain constant throughout the process, the shading effect of turbulizer on absorber plate is ignored, heat transfer between turbulizer and absorber plate is disregarded, and the influence of measuring instruments in the flow field is neglected.. In accordance with the above assumptions, heat loss in SAC can be calculated as the sum of convective and radiative heat exchanges between the transparent coating and the external environment.
- **Thermal-technical parameters of the turbulizer SAC:** The three- and two-dimensional geometry of the studied turbulizer SAC is shown in Figure 1. The sac consists of five main components: a transparent coating, an absorber plate, a turbulizer, the sac body, and a transparent coating. There is also an air duct located between the absorber plate. The dimensions of the SAC are as follows: length 1,000 cm, width 500 nm, and thickness 60 m. A 50-mm diameter pipe is installed at the entrance and exit sections of the SAC. The geometric dimensions of turbulizers installed on the absorber plate of the sac are: 40 m long, 4 in number, and 5 m high. The height of all turbulizers completely occupies the entire height of the air duct, which is 80% of the cross-sectional area of air. Turbulizers were made from 2-mm thick metal plates attached to the absorber plate and sac body by electric welding. They were painted black.. The absorber plate and sidewalls of SAC are made from a 2 mm thick metal sheet and painted black to maximize the absorption of solar radiation energy. A single layer of glass, 4 mm thick, was used as the transparent coating for SAC. The absorber plate and back of the sidewalls are insulated with 20 mm thick polystyrene. The outer surface of the insulation is protected from external influences by a non-ferrous metal sheet, 1 mm thick [15].

In the proposed SAC with turbulizer, it is necessary to develop physical and mathematical models of SAC with turbulizer to determine the flow structure, velocity and pressure loss in order to study the influence of the turbulizer shape, the number of turbulizers and the air flow velocity at the inlet on the flow characteristics of the SAC. In this case, the fluid (air) flow and its properties are assumed to be constant. Air is used as the working medium. The Prandtl number is 0.71, and the air flow velocity varies in the range of 4…20 m/s. At the entrance to the channel, a uniformly distributed one-dimensional velocity is used as the aerodynamic boundary condition [16,17]. Also, the boundary conditions of no-slip and no-flow are given to the walls, and atmospheric pressure (P=1 atm) is assumed at the exit from the channel [16]. Since the air flow in the SAC is close to a rectangular flow, the internal air flow can be considered as a turbulent flow. Taking into account the strong flow discontinuity and buoyancy force, the k-ε model was chosen as the turbulence model.

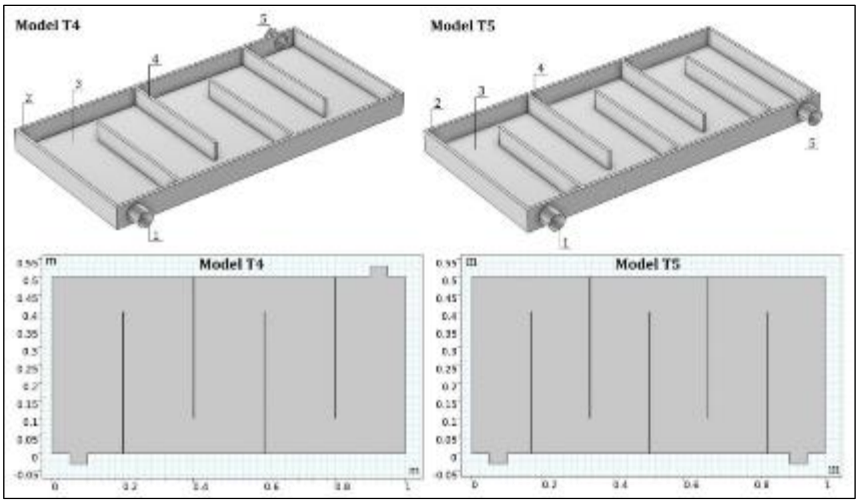

Figure 1 Three-and two-dimensional (3D and 2D models) geometries of the solar air collector with turbulizer: 1, 5-air transmission and exhaust pipes; 2-side walls; 3-absorber Plate; 4-turbulizer

3. Results and discussion

The number of models was developed using the COMSOL Multiphysics 6.1 software. This software is based on the finite element method and is widely used for numerical modeling of hydrodynamics, turbulence, and heat and mass exchange processes. Using the COMSOL tool, we initially researched the characteristics of internal flow in a turbulizer sac on a 2D basis. In the T4 and T5 models, we showed the image of flow lines that characterize the internal flow of air in Figure 2. As can be seen from the flow lines, the installation of turbulizers leads to a more complex flow structure. In these models, large-scale shocks form after the turbulizer, causing strong current separation and re-coupling.. At the end of the turbulizing element, the air stream turns at an angle of 180 degrees, and on the back side of the device, large longitudinal and localized bumps are formed, with a sufficient margin. Additionally, dormant zones of airflow are created. Changes in the velocity of the airflow in these models do not significantly alter the flow characteristics.

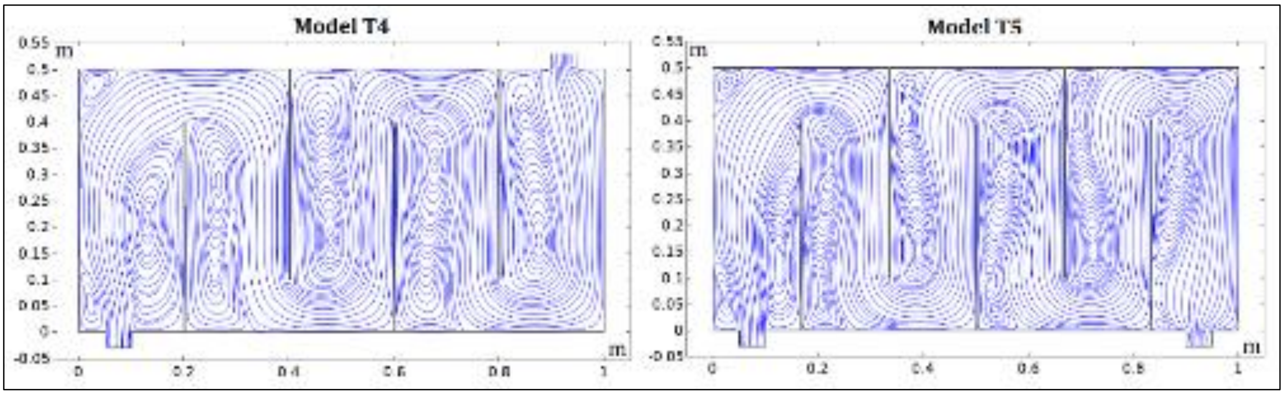

Figure 2 T4 va T5 modellarda oqimni xarakterlovchi oqim liniyalari

The change in velocity along the length and width of the SAC at different air flow rates in the T4 and T5 models is shown in the form of a diagram and a graph in Figure 3. As can be seen from the results, the velocity is high in front of the turbulizer, and very low in the back, and the velocity increases to a maximum value due to the reduction in the cross-section of the air flow when bypassing the turbulizer. The decrease in velocity to a minimum value behind the turbulizers is due to the presence of a recirculation zone. In both T4 and T5 models, the velocity change pattern is the same, with a significant decrease in velocity after each turbulizer. The change in velocity along different widths of the SAC is also almost the same. The sharp increase and sharp decrease in velocity in each chamber is associated with the narrowing and expansion of the cross-section of the air flow channel. The velocity values are uneven in the first and second chambers, but evenly distributed in the remaining chambers, which indicates that the flow is moving uniformly, as a result of which the entire surface of the SAC does not fully participate in heat exchange.

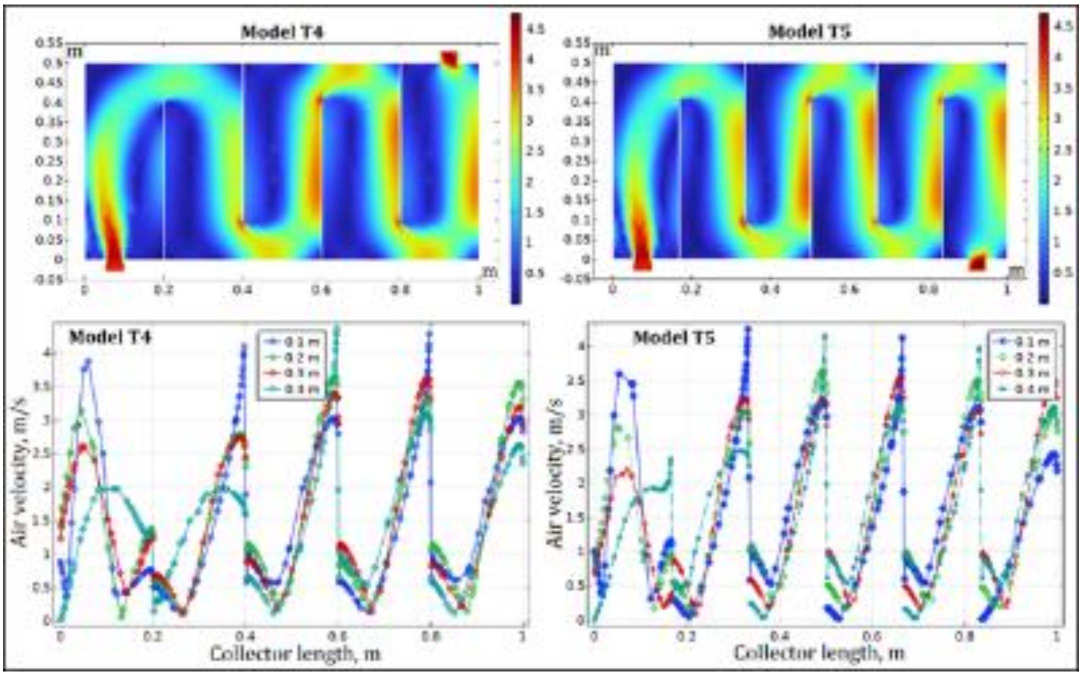

Figure 3 Results of flow rate changes in T4 and T5 models

The variation of air flow pressure along the length and width of the SAC in the T4 and T5 models is shown in diagram and graphic form in Figure 4.

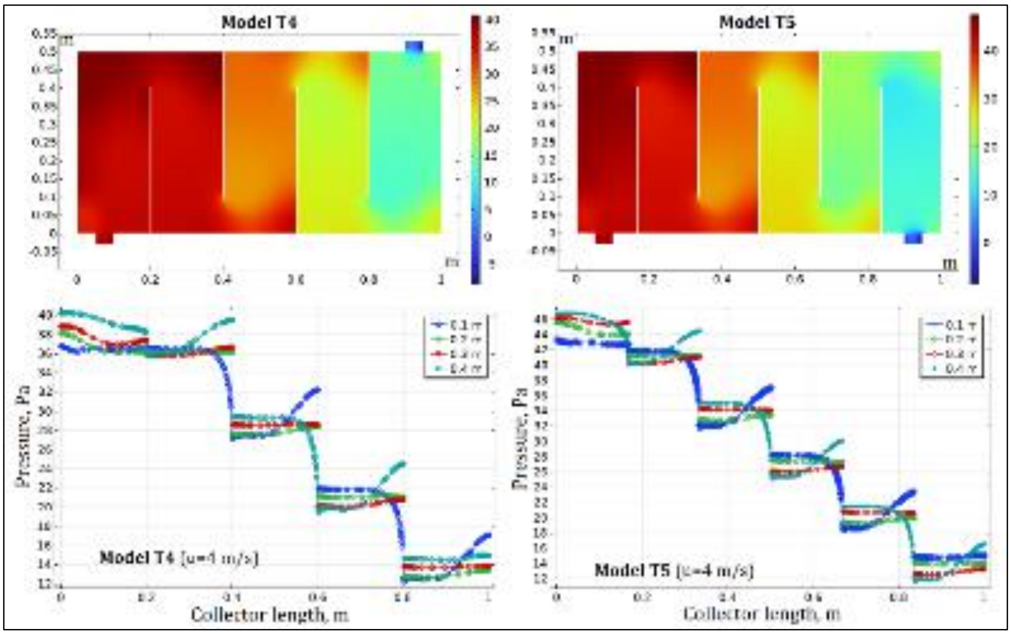

Figure 4 Results of pressure changes in models T4 and T5

From the results, it can be seen that the change in airflow pressure in the T4 model was directly dependent on speed, and the pressure was decreasing as the airflow speed increased. After passing through each turbulizer, the pressure of the air flow has decreased significantly. The pressure difference was Δp=28 Pa when the flow rate was u=4 M/s. Due to the increase in the number of turbulizers in the T5 model, the air pressure at the entrance increased by about 1.15 times. The T4 and T5 models had nearly the same output pressure value. The pressure difference was Δp=34 Pa when the flow rate was u=4 M/s. In both models, the character of the change in pressure over SAC width was almost identical.

4. Conclusion

The following conclusions were made on the numerical study of the flow characteristic in turbulizer SAC:

Physical models of turbulizer sacs have been developed that allow the acceleration of thermal-hydrodynamic processes in sacs. A model has been developed that allows a complete study of hydrodynamic processes in turbulizer SAC and is based on the K-ε model of turbulence in the COMSOL Multiphysics 6.1 program. When the characteristic of airflow was numerically researched in SAC's developed T4 and T5 models, it was found that the distribution of airflow over the surface of the SAC absorber was nearly identical. When the velocity of airflow in a turbulizer SAC is numerically researched at u=4 m/s, the mean velocity of flow in the air duct is 3-3.5 m/s,in these models the maximum difference in pressures over SAC length is 28...34 PA

Compliance with ethical standards

Disclosure of conflict of interest

No conflict of interest to be disclosed.

References

[1] Liang C.H., Zhang X.S., Li X.W., Zhu X. Study on the performance of assisted air source heat pump system for building heating. Energy Build. 43, 2011. – p. 2188-2196.

[2] Abedi A. Utilization of solar air collectors for heating of heating of Isfahan buildings in Iran. Energy Proc. 14, 2012. – p. 1509-1514.

[3] Ibragimov U.Kh., Kholmatov F.T., Avanesov T.R., Botyrov A.S. Methods of using solar air collectors in air heating and cooling systems of buildings // Problems of architecture and construction, 4, 2023. – p. 273-275.

[4] Karsli S. Performance analysis of new-design solar air collectors for drying applications. Renew. Energy 32, 2007. – p. 1645-1660.

[5] Ayadi M., Mabrouk S.B., Zouari I., Bellagi A. Simulation and performance of a solar air collector and a storage system for a drying unit. Sol. Energy 107, 2014. – p. 292-304.

[6] Gulcimen F., Karakaya H., Durmus A. Drying of sweet basil with solar air collectors. Renew. Energy 93, 2016. – p. 77-86.

[7] Bhattacharyya T., Anandalakshmi R., Srinivasanc K. Heat transfer analysis on finned plate air heating solar collector for its application in paddy drying. Energy Proc. 109, 2017. – p. 353-360.

[8] Ibragimov U.Kh. Analysis of the current state of the use of solar air collectors in the world and their designs // Problems of Architecture and Construction, 1, 2024. – p. 222-277.

[9] Velmurugan P., Ramesh P. Evaluation of Thermal Performance of Wire Mesh Solar Air Heater. Indian Jour. of Science and Technology, 4, 2011, - p. 12-14.

[10] Yeh H.M., Ho C.D., Lin C.Y. The Influence of Collector Aspect Ratio on the Collector Efficiency of Baffled Solar Air Heaters. Energy, 23, 1998. – p. 11-16.

[11] Moummi N., Youcef-Ali S., Moummi A., Desmons J.Y. Energy analysis of a solar air collector with rows of fins. Renew. Energy 29, 2004. – p. 2053-2064.

[12] Bensaci C.E., Moummi A., Labed A. Experimental investigation on heat transfer coefficient and thermal efficiency of solar air heaters having different baffles Green Energy and Technology (Springer International Publishing), 2020. – p. 309-332.

[13] Romdhane B.S. The air solar collectors: comparative study, introduction of baffles to favor the heat transfer. Sol. Energy 81, 2007. – p. 139-149.

[14] Oshovsky V.V., Okhrimenko D.I., Sysoev A.Yu. Use of computer systems of finite element analysis for modeling hydrodynamic processes // Scientific Proceedings of DonNTU, 15(163), 2010. – p. 163-173.

[15] Ibragimov U.Kh., Kadirov J.R. Numerical study of the turbulence effect in a solar air collector with a barrier turbulizer // Scientific and Technical Journal of the Fergana Polytechnic Institute, 5, 2024. – p. 121-128.

[16] Demartini L.C., Vielmo H.A., Möller S.V. Numeric and experimental analysis of the turbulent flow through a channel with baffle plates. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 26(2), 2004. – p. 153-159.

[17] Nasiruddin Kamran Siddiqui M.K. Heat transfer augmentation in a heat exchanger tube using a baffle. International Journal of Heat and Fluid Flow, 28(2), 2007. – p. 318-328.