Jurnal Tekno Insentif | Vol. 17 | No. 2 | Oktober 2023
DOI : https://doi.org/10.36787/jti.v17i2.1172 Halaman 150-159

Computational Analysis of Horizontally and Inclined Finned Solar Air Collector

HALEFOM KIDANE[1], JANOS BUZAS[2], ISTVAN FARKAS[2]

[1]Doctoral School of Mechanical Engineering, Hungarian University of Agriculture and Life Sciences, Hungary
[2]Institute of Technology, Hungarian University of Agriculture and Life Sciences, Hungary
Email: Abrha.Halefom.Kidane@phd.uni-mate.hu

ABSTRACT

The majority of researchers have carried out simulations using computational fluid dynamics (CFD) and experimental investigation for design, manufacturing, performance evaluation, understanding flow behavior, efficiency prediction, condition optimization, etc. The main objective of the recent paper is to compare horizontally finned and 45° inclined finned solar air collector absorbers and to select the better construction using numerical simulation. ANSYS Fluent Workbench 2022 with renormalization-group (RNG) group k–ε turbulence model was applied for the analysis of the computational domain of solar air heaters. The simulation results generated using CFD were validated with data from the literature, and the simulation results were in agreement with experimental results from the relevant literature. Based on the simulation results the horizontal finned solar air collector absorber has better heat transfer behavior than the 45° inclined.

Keywords: computational fluid dynamics, finned absorber, renormalization-group, simulation, solar air heater

ABSTRAK

Mayoritas peneliti umumnya melakukan simulasi menggunakan perangkat komputasi, seperti computational fluid dynamic (CFD) dan eksperimental untuk menunjang keperluan desain, rancang bangun sistem, evaluasi kinerja dan karakteristik aliran, memperkirakan kinerja, serta optimalisasi, dll. Target dari makalah ini adalah membandingkan pemanas udara surya dengan sirip posisi horizontal dan miring 45° yang dipasang pada kolektor surya untuk memilih konstruksi terbaik secara numerik. ANSYS Fluent Workbench 2022 dengan model turbulensi k–ε renormalisasi grup (RNG) digunakan sebagai perangkat simulasi. Hasil simulasi divalidasi dengan data eksperimental dari literatur yang relevan, dan memperlihatkan kecenderungan yang sama. Berdasarkan simulasi, kolektor udara surya bersirip horizontal mempunyai karakteristik perpindahan panas yang lebih baik dibandingkan dengan kolektor udara bersirip miring 45°.

***Kata kunci**: komputasi dinamika fluida, kolektor bersirip, renormalisasi grup, simulasi, pemanas udara surya air*

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is one of the most powerful tools used by many researchers in different research areas because it is a fast and low-cost technique for solving different engineering problems, involving fluid flows and heat transfer. The system designers and analysts now have a tool called CFD which is cost-effective than experiments and produces more data, including data that is not yet quantifiable with available instruments **(Rodriguez, 2019)**. In addition, a CFD enables analysts and designers to investigate deeper and farther, into the specifics of system behaviors.

As briefly outlined, the best method is always to use CFD models that have been verified through experimental research for further design and optimization goals **(Duta et al., 2020)**. A recent tendency favors defining the entire process by reducing the simplifications on the cost of computational time and cost. Presently, certain assumptions and approximations in the computational geometry and boundary conditions of the process were favored to lower the computational time.

Several experimental and numerical investigations used CFD, to model and simulate solar air heaters for different purposes. For example, **Badache et al. (2013)** constructed a two-dimensional, unglazed transpired solar air collector and performed an experiment and numerical simulation. Thus, their finding reveals that the CFD simulation and experimental findings were in good agreement. **Karmare et al. (2010)** applied computational fluid dynamics and experimental methods to analyze the fluid flow and heat transfer in a solar air heater with a rib-grit roughened surface. It was discovered that there is good agreement between experimental and CFD analytical results. **Potgieter et al. (2020)** studied the thermal analysis and experimental research of a solar air heater with rectangular rib roughness on the absorber plate have been demonstrated. The simulated minimum and maximum temperatures of the solar air heater were reasonably in line with the experimental results. **Komolafe et al. (2019)** conducted CFD-based correlations and numerical simulation for a synthetically roughened solar air heater. There was a 10% difference between the CFD result and the experimental data set. **Moradi et al. (2017)** designed a novel solar air heater elaborated with the support of CFD simulation and tested the sample experimentally. They obtained an agreement between the results of the numerical simulation and experimental results.

The analysis of characteristics and roughness was conducted experimentally and a computational method was applied to forecast the flow dynamics across the heat-collector plate. The values derived from the developed correlation and experimental values were closely fitted **(Ingle et al., 2008). Korpale et al. (2020)** studied the solar air heater optimization and numerical simulations. CFD models and experiments have been used to confirm the ideal design parameter values. The accuracy of the empirical correlations utilized for the design of the solar artificial air heater and the appropriate selection of model equations were demonstrated by the errors being within an acceptable range**. Kumar et al. (2017)** used CFD to characterize the fluid flow properties and heat transfer in a ribbed triangular duct. A good agreement between the numerical outcomes and the available results is found, with a maximum error of ±4.04%. **Gawande et al. (2016)** investigated Convection heat transmission in a solar air heater with reverse L-shaped ribs experimentally and via ANSYS Fluent CFD code. From the variety of parameters examined, it has been discovered that the numerical findings and the experimental data were in good agreement**. Amraoui et al. (2015)** compared the theoretical and the experimental results with utilising a computational fluid dynamics tool in terms of flow and temperature distribution inside the solar collector.

2. METHODS

The solar air heater considered here is found in Hungarian University of Agriculture and Life Sciences (MATE), Gödöllő, Hungary at the place called Solar Energy Laboratory. It is located at a latitude of 47.59° N, longitude of 19.36° E as shown in Fig. 1. The site is characterized with warm summers, freezing and snowy winters and partly cloudy year-round. Over the course of the year, the temperature typically varies from -4.44 °C to 26.67 °C and is rarely below -12.22 °C or above 32.22 °C.

Figure 1. The location of the laboratory from Google earth

The 3D model shown in Fig. 2 and Fig. 3 of the horizontal and finned solar air heater geometries were drawn using SOLIDWORKS 2022 version with dimension of 121.8 cm × 46.2 cm (L × W) respectively for both types and the numerical simulations were performed in ANSYS Fluent 2022 R1.

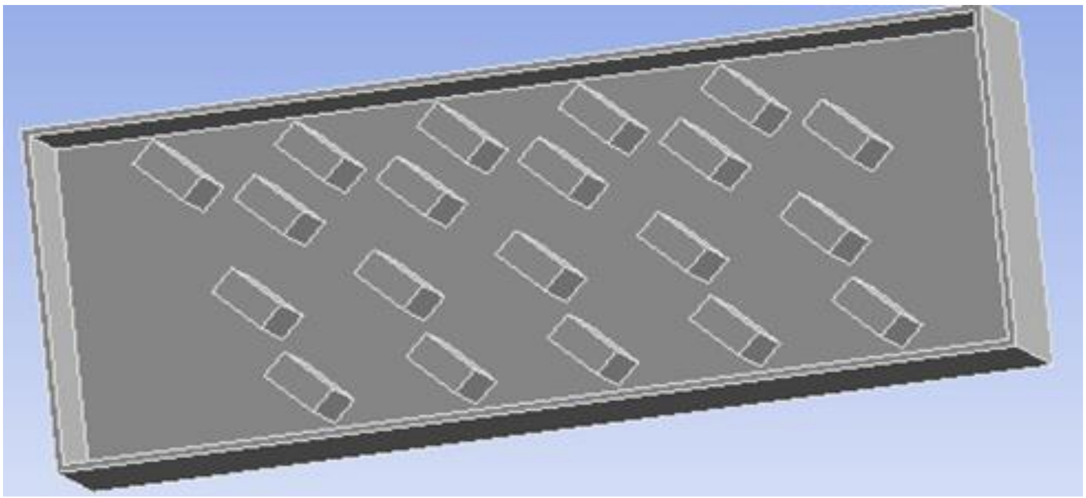

Figure 2. 45° inclined finned absorbing surfaces of solar air heater

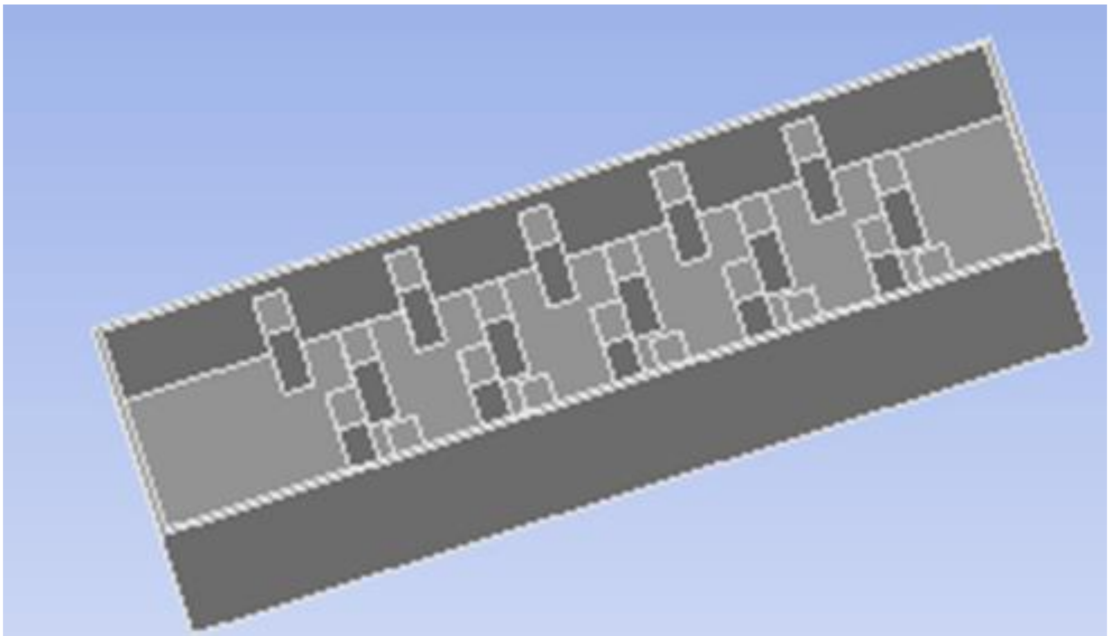

Figure 3. Horizontally finned absorbing surfaces of solar air heater

The default meshing which is coarse mesh (Fig. 4) is applied in solar air heater parts including the baffles or finned to analyses the research of stream and heat transmission and other simulation parameters.

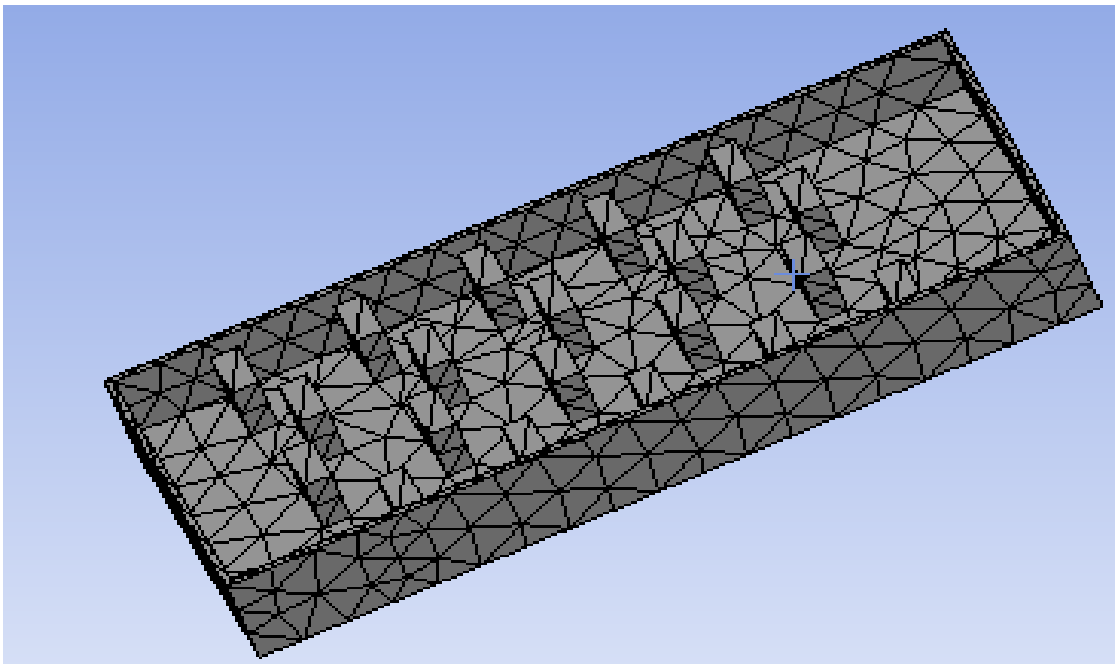

Figure 4. Meshing the horizontally finned absorbing surfaces of solar air heater

Initial condition: The working fluid which air is come from surround and enter to the solar air heater, where it is heated by sun ray inside the solar air heater. So, such setup was used as initial conditions for the simulation.

Boundary conditions: No slip wall condition was used.

Inlet: The outlet air velocity and temperature of the flat plate collector were taken from previously conducted experiments in the stated laboratory **(Al-Neama et al., 2018)**, and used as an inlet to the drying chamber. The turbulence intensity was set at 10%. The detail of initial and boundary conditions as well as the other properties of the computational system are given in Table 1.

Table 1. Initial and boundary conditions used during the simulation

Parameter	Value
Density of air (kg/m^3)	1.2
Specific heat capacity of air (J/kg K)	1007
Walls	Adiabatic
Outlet	Assumed gauge pressure
Inlet, outlet air temperature and inlet air velocity	294.45 K, 310.45 K and 2.3 m/s
Global solar insolation intensity	946 W/m^2

The main governed equation solved here are continuity, momentum and energy equations are written as follow **(Anand et al., 2020)**:

Continuity equation:

$$\partial/\partial x_i(\rho u_i) = 0, \tag{1}$$

where, ρ is density (kg/m^3) u is air velocity (m/s).

Momentum equation:

$$\partial/\partial x_i(\rho u_i u_j) = -\partial p/\partial x_i + \partial/\partial x_j[\mu(\partial u_i/\partial x_j + \partial u_j/\partial x_i)] + \partial/\partial x_j\left(-\overline{\rho(u_i)u_j}\right), \tag{2}$$

Where, p is the pressure, μ is the shear viscosity.

Energy equation:

$$\partial/\,\partial x_i(\rho u_i T) = \partial/\,\partial x_j\left((\Gamma + \Gamma_t)\ \partial T/\,\partial x_j\right), \tag{3}$$

$$p/\rho = RT, \tag{4}$$

where, T is the temperature, Γ and Γ_t are molecular thermal diffusivity and turbulent thermal diffusivity, respectively and are given by $\Gamma = \mu/Pr$, and $\Gamma_t = \mu_t/Pr_t$, where Pr is the Prandtl number, and Pr_t is the turbulent Prandtl number, R is the characteristic gas constant = 0.287 kJkg^{-1}K^{-1} and ρ is a function of temperature.

.

3. RESULTS AND DISCUSSION

The simulation results as the static temperature distribution, the turbulence kinetic energy and the pressure contour are discussed for the horizontally finned and 45° inclined finned solar air collector absorbers. As shown in Fig. 5, in the case of the 45° inclined finned absorber the inlet air temperature rises from 294 K to 318 K at a steady state. Under the set boundary conditions, the input temperature was set as 294 K and the output temperature 310 K. Thus, the fins have a positive effect on heat transfer. As shown in Figs 5 and 6, the outlet part with red color arrows has high outlet air temperatures. Comparing the simulation results, the horizontal fins is slightly better than the 45° inclined finned. The result was validated using the experimental work **(Al-Neama et al., 2019)**. The simulation results agreed with experimental values.

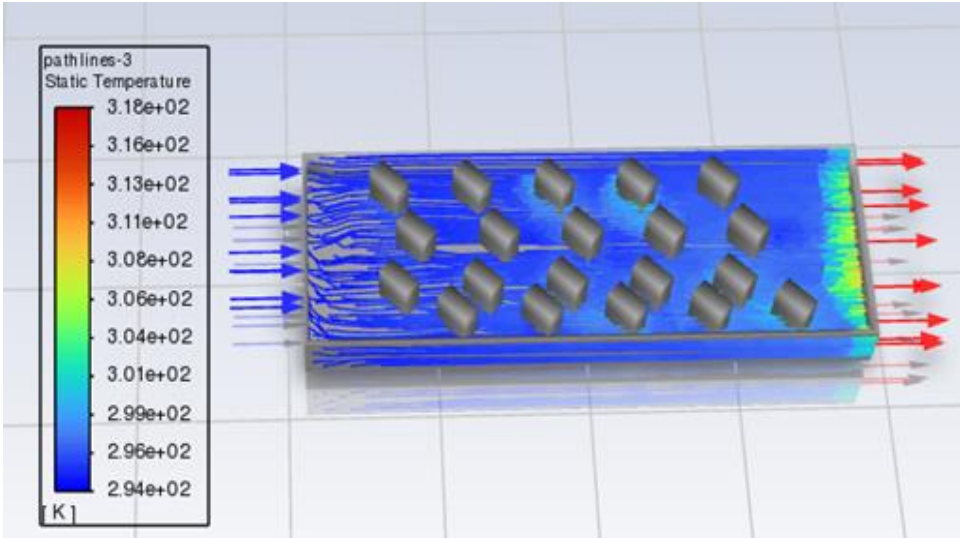

Figure 5. Inclined (45°) finned absorbing surfaces of solar air heater temperature distribution

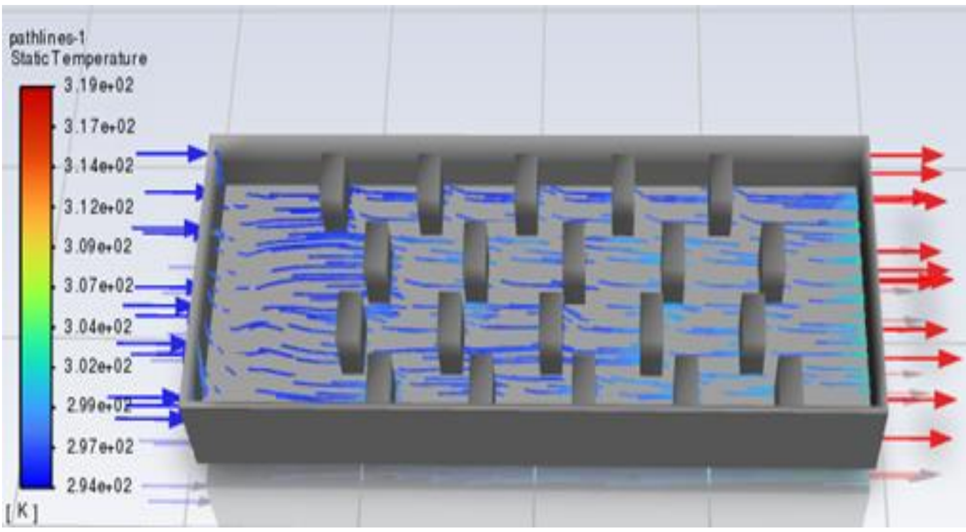

Figure 6. Horizontally finned absorbing surfaces of solar air heater temperatures streamlines

As stated by **Phu et al. (2021)** the baffle in the air collector duct increases heat transfer, while also significantly increasing the pressure loss penalty. In other words, the baffle added to solar air heater accelerates heat transmission simultaneously they drop sharply raising the cost of pressure loss (Fig. 7).

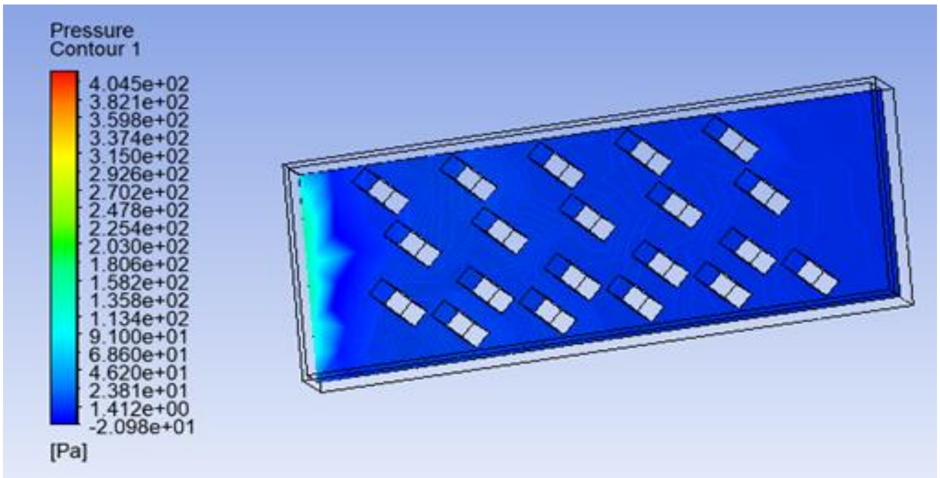

Figure 7. Inclined (45°) finned absorbing surfaces pressure contour

The heat transfer phenomenon can be observed and described by the contour of turbulence kinetic energy. The contour plot of turbulence kinetic energy is shown in Fig. 8 and Fig. 9 for both cases. The value of turbulence kinetic energy of 45° inclined finned absorbing surfaces varies between 0.00382 and 1.615 m2s-2 (see Fig. 8).

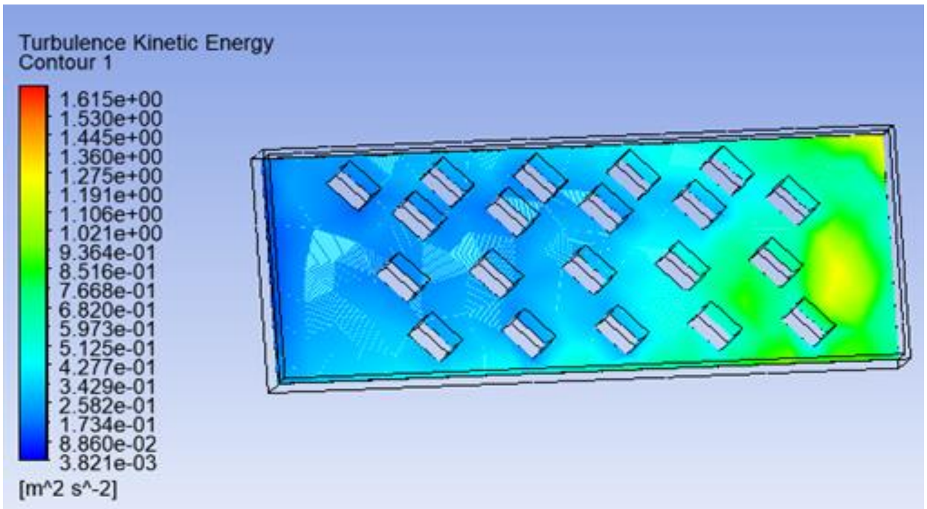

Figure 8. Inclined (45°) finned absorbing surfaces turbulence kinetic energy contour

As shown above in Fig. 9 the value of turbulence kinetic energy of horizontally finned absorbing surfaces varies between 0.00614 and 2.87 m^2s^{-2}. So, we can roughly say the horizontally finned surface of solar air heater has better heat transmission approach.

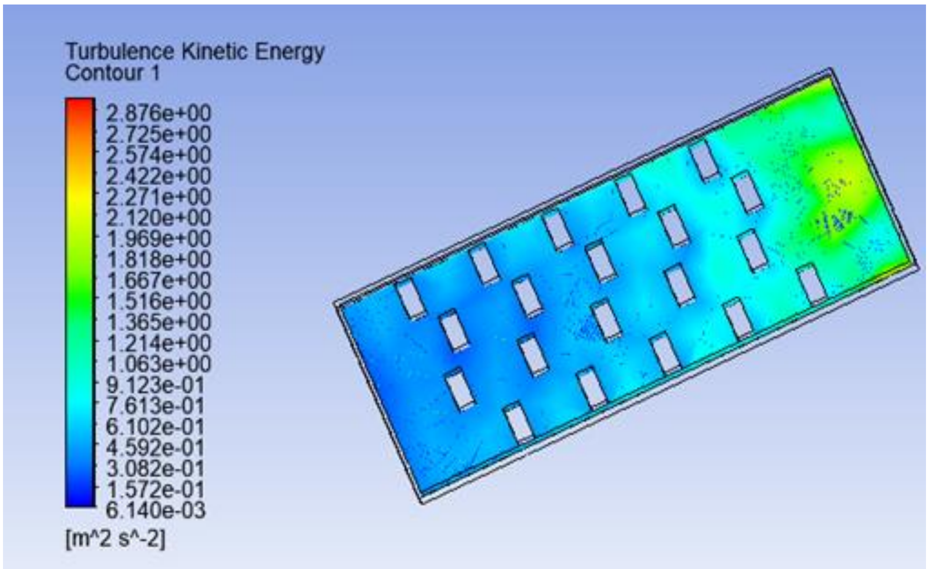

Figure 9. Horizontally finned absorbing surfaces turbulence kinetic energy contour

4. CONCLUSIONS

A computer simulation approach called computational fluid dynamics (CFD) is used to carry out a broader range of activities such as heat and mass transfer, chemical reactions, and biological activities. Computational fluid dynamics is better than experimental investigation because it takes a short time to run the simulation compared to the experiment, it's low cost, uses any scale, it's repeatable, and it's secure. Solar air heater is one of the basic equipment through which solar energy is converted into thermal energy. Numerical investigation of horizontally finned and 45° inclined finned solar air collector absorbers have been carried out. The horizontally finned solar air heater has better heat transfer properties than the inclined finned.

ACKNOWLEDGEMENTS

This work was supported by the Stipendium Hungaricum Programme and the Doctoral School of Mechanical Engineering, Hungarian University of Agriculture and Life Sciences (MATE), Gödöllő, Hungary.

REFERENCES

Al-Neama, M.A. Performance enhancement of solar air collectors applied for drying processes. Ph.D. Thesis. Szent István University, Gödöllő, Hungary; 2018.

Al-Neama MA, Farkas I. Thermal efficiency of vertical and horizontal-finned solar collector integrated with forced air circulation dryer for Apple as a sample. Drying Technology, 2019; 37(5):546–558. doi: https://doi.org/10.1080/07373937.2018.1488260.

Amraoui MA, Aliane K. Numerical Study of the Three-dimensional Flow in a Flat Plate Solar Collector with Baffles. CFM 2015 - 22ème Congrès Français de Mécanique; 2015 Aug 24-28; Lyon, France. Available from: https://hal.archives-ouvertes.fr/hal-03444952

Anand S, Mishra DP, Sarangi SK. CFD supported performance analysis of an innovative biomass dryer. Renew Energy, 2020 Oct; 159:860–872. doi: https://doi.org/10.1016/j.renene.2020.06.039

Badache M, Rousse, DR, Hallé S, Quesada G. Experimental and numerical simulation of a two-dimensional unglazed transpired solar air collector. Sol Energy. 2013 Jul; 93:209–219. doi: https://doi.org/10.1016/j.solener.2013.02.036.

Dutta A, Erdoğdu F, Sarghini F. Computational Fluid Dynamics (CFD) Simulations in Food Processing. Mathematical and Statistical Applications in Food Engineering, 1st ed. Sevda S, Singh A. Ed. Boca Raton: CRC Press, 2020, pp. 244–261. doi: https://doi.org/10.1201/9780429436963.

Gawande VB, Dhoble AS, Zodpe DB, Chamoli S. Experimental and CFD investigation of convection heat transfer in solar air heater with reverse L-shaped ribs. Sol Energy. 2016 Jun; 131:275–295. doi: https://doi.org/10.1016/j.solener.2016.02.040.

Ingle PW, Pawar AA, Deshmukh BD, Bhosale KC. CFD Analysis of Solar Flat Plate Collector. International Journal of Emerging Technology and Advanced Engineering. 2008. Available from: www.ijetae.com.

Karmare SV, Tikekar AN. Analysis of fluid flow and heat transfer in a rib grit roughened surface solar air heater using CFD. Sol Energy. 2010 Mar; 84(3): 409–417. doi: https://doi.org/10.1016/j.solener.2009.12.011.

Komolafe CA, Oluwaleye IO, Awogbemi O, Osueke CO. Experimental investigation and thermal analysis of solar air heater having rectangular rib roughness on the absorber plate. Case Studies in Thermal Engineering. 2019 Sep; 14. doi: https://doi.org/10.1016/j.csite.2019.100442

Korpale VS, Deshmukh SP, Mathpati CS, Dalvi VH. Numerical simulations and optimization of solar air heaters. Appl Therm Eng. 2020 Nov 5; 180. doi: https://doi.org/10.1016/j.applthermaleng.2020.115744.

Kumar R, Kumar A, Goel V. A parametric analysis of rectangular rib roughened triangular duct solar air heater using computational fluid dynamics. Sol Energy. 2017 Nov 15; 157:1095–1107. doi: https://doi.org/10.1016/j.solener.2017.08.071.

Moradi R, Kianifar A, Wongwises S. Optimization of a solar air heater with phase change materials: Experimental and numerical study. Exp Therm Fluid Sci. 2017 Dec; 89:41–49. doi: https://doi.org/10.1016/j.expthermflusci.2017.07.011.

Phu NM, Thao PB, Hap NV. Effective efficiency assessment of a solar air heater having baffles spaced with different successive ratios. Case Studies in Thermal Engineering. 2021 Sept 25; 28:101486. doi: https://doi.org/10.1016/j.csite.2021.101486.

Potgieter MSW, Bester CR, Bhamjee M. Experimental and CFD investigation of a hybrid solar air heater. Sol Energy. 2020 Jan 1; 195:413–428. doi: https://doi.org/10.1016/j.solener.2019.11.058

Rodriguez S. Applied computational fuid dynamics and turbulence modeling, Practical Tools, Tips and Techniques. First edition. Cham: Springer; 2019. doi: https://doi.org/10.1007/978-3-030-28691-0.