INVESTIGATION OF HEAT AND MASS EXCHANGE PROCESSES OF HELIUM DRYER OPERATION BY ANALYTICAL METHOD AND USING CFD ANALYSIS IN SOLIDWORKS FLOW SIMULATION ENVIRONMENT

Serhiy Korobka[1], **Mykola Tolstushko**[2], **Igor Stukalets**[1], **Viktor Sheichenko**[3],
Igor Kryvovyazyuk[2], **Nataliya Tolstushko**[2]
[1]Lviv National Environmental University, Ukraine; [2]Lutsk National Technical University, Ukraine;
[3]Poltava State Agrarian University, Ukraine
korobkasv@ukr.net, tmmtno@gmail.com, igorstukalets@gmail.com, vsheychenko@ukr.net,
krivovyazukigor@gmail.com, nataleksa1978@gmail.com

Abstract. The operation of solar dryers is based on the conversion of solar radiation into thermal energy using air collectors. To stabilize the technological process in conditions of low insolation, the structures are equipped with heat accumulators and heat pump units. This allows for energy independence and preserves the biological value of the raw materials. A new helium dryer design with a sealed and insulated body and a flat absorber has been developed, which can be used as an additional heating element of a low-temperature heat source. Generalizing dependences for determination of heat efficiency of the helio-dryer, in particular the influence of components of heat balance of the installation and level of insolation on temperature difference of flows of heat carrier in the collector and heat productivity are determined. In the SolidWorks CAD environment, a solid-state design model of the helio-dryer was created, and a computer simulation of its operation was performed. Using CFD analysis in SolidWorks Flow Simulation, the process of heating the absorber plate under the influence of solar radiation was simulated and temperature distribution diagrams were obtained, the heat transfer coefficient, temperature and air flow velocity were determined, and heat flow patterns in the working area of the solar dryer were visualized. The distribution of the temperature field along the absorption panel was obtained, which made it possible to improve the mathematical model of the heat exchange process in the developed installation.

Keywords: solar collector, air heating system, absorber, heat flow, Rayleigh criterion, heat efficiency.

Introduction

The air solar collector of the helio-dryer is a heat-receiving module consisting of a transparent coating, an absorber, an air channel, thermal insulation and a hermetic housing. It is designed to convert solar radiation into thermal energy and then transmit it to the air flow. The structural diagram of the air solar collector of the solar dryer is shown in Fig. 1.

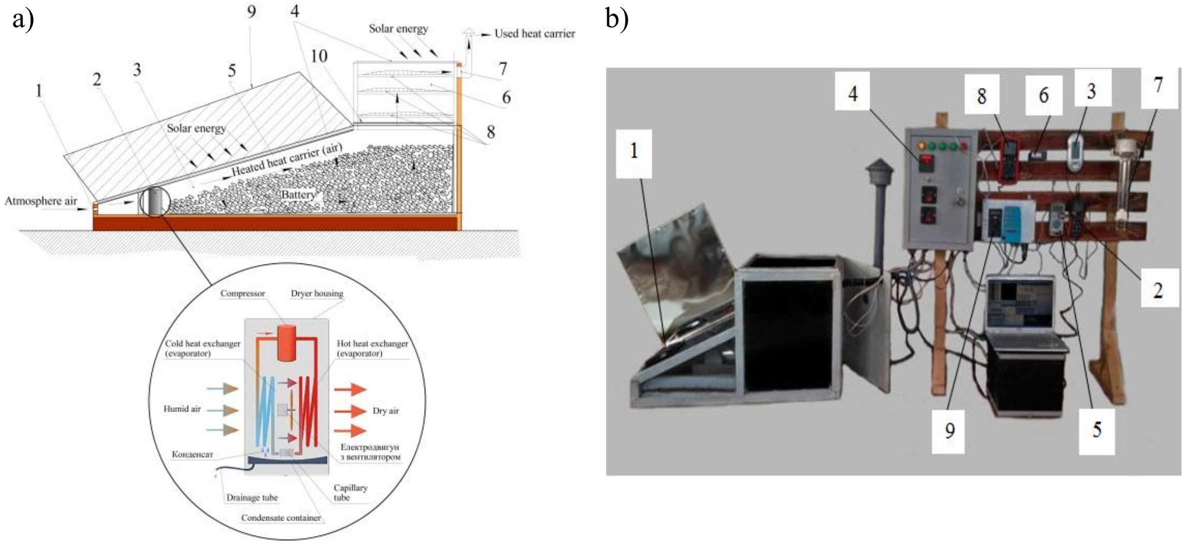


Fig. 1. **Structural and technological scheme of the helio dryer with a thermal accumulator and flat mirror concentrator (a) and a general view of the experimental installation (b):** 1 – input channel; 2 – coolant moisture dryer; 3 – air duct; 4 – air manifold; 5 – heat accumulating material (TAM); 6 – drying chamber; 7 – exhaust duct; 8 – sieve; 9 – flat mirror concentrator; 10 – flap [1]

Solar radiation passes through a transparent coating, heats the absorber (up to 60-90 ºC), after which heat is transferred to the air in the channel by convective heat exchange The efficiency of the collector is 40-70%, and the temperature of the outlet air usually exceeds the ambient temperature by 20-40 ºC under conditions of natural or forced convection. Determination and justification of the thermal parameters of the air heliocollector is an important stage in the design of effective solar heating and air heating systems. Rational selection of the characteristics of the collector allows increasing its thermal output and ensuring stable operation in various climatic conditions [1-10].

Materials and methods

The main indicators of the operation of the solar collector are the useful heat productivity [2-4] per unit area q_u [5] and efficiency, which is determined by the efficiency factor quantitatively η [6]. They are described by the classical equations of the energy balance:

$$q_u = F_R \cdot \left(\eta_0 \cdot E - U_L \cdot (t_{at} - t_a) \right) , \tag{1}$$

$$\eta = F_R \cdot \left(\eta_0 - U_L \cdot \left((t_{at} - t_a) / E \right) \right), \tag{2}$$

where $F_R = \left(G_m \cdot c_p / U_L \right) \cdot \left(1 - e^{-F' \cdot U_L / \left(G_m \cdot c_p \right)} \right)$ – heat removal coefficient (flow rate) of the solar collector [7; 8];
G_m – mass flow rate of coolant;
c_p – specific heat of the coolant [9];
U_L – coefficient of heat loss environment for the collector;
F – efficiency coefficient of the collector's absorbing surface;
t_{at} – coolant temperature;
t_a – ambient temperature;
E – incoming solar radiation of the receiving surface;
η_0 – optical efficiency of the air collector.

The temperature of the panel increases along the flow at the flow air manifolds. The efficiency factor of the slot collector is determined as the ratio of the useful heat flow to the total, taking into account the heat loss:

$$F' = q_u / \left(q_u + q_{mb} \right). \tag{3}$$

where q_u – useful specific (per unit area) heat productivity of the collector;
q_{mb} – useful specific (per unit area) heat productivity of the coolant in the heliodryer.

The efficiency factor of the air collector depends on the flow rate of the coolant and the purpose of the system: at a high flow rate, it approaches one, providing optimal heat productivity, and for dryers the emphasis is placed on increasing the temperature of the outlet stream due to the regulation of the collector mode [1; 10].

The efficiency of the air collector as part of the dryer is estimated by the heat productivity and output flow temperature, which functionally depend on the kinetic and thermophysical parameters of the input values and the system of heat balance equations on the key elements of the structure; for heat transfer analysis, the CFD (Computational Fluid Dynamics) method is used in SolidWorks, namely as follows.

- The thermal balance of the glass coating takes into account the three inputs (solar radiation $\alpha_g \cdot E$, panel radiation $\alpha_{rpg} \cdot (T_p - T_g)$ and convection $\alpha_{cpg} \cdot (T_p - T_g)$ and two outflows (convection with wind V_w ($\alpha_w = 5.7 + 3.8 V_w$) and radiation losses); when neglecting absorption in glass, its equation is simplified:

$$\alpha_g \cdot E + \left(\alpha_{cpg} + \alpha_{rpg} \right) \cdot \left(T_p - T_g \right) = \left(\alpha_w + \alpha_{rga} \right) \cdot \left(T_g - T_a \right). \tag{4}$$

where T_p – air collector panel (absorber) temperature;
T_g – temperature of the transparent glass cover of the air collector.

- Heat balance of the absorption panel $E \cdot \eta_0$ takes into account one input stream of solar energy $E \cdot \eta_0$ and four output (convection $\alpha_{cpg} \cdot (T_p - T_g)$ and radiation to glass $\alpha_{rpg} \cdot (T_p - T_g)$ and coolant

$\alpha_{cpf}\cdot(T_p - T_f)$, at that coolant temperature $T_f = 0{,}5\cdot(T_p + T_g)$ is taken as average between inlet and outlet:

$$\eta_0 \cdot E = \left(\alpha_{cpg} + \alpha_{rpg}\right)\cdot\left(T_p - T_g\right) + \alpha_{cpf}\cdot\left(T_p - T_f\right) + \alpha_{rpb}\cdot\left(T_p - T_b\right), \tag{5}$$

where T_p – absorber (panel) temperature;
T_g – glass coating temperature;
T_f – average temperature of the coolant (liquid in the collector).

- Absorption panel transfers power to the coolant $\alpha_{cpf}(T_p - T_f)$, most of which forms a useful heat flow $G_m\cdot c_p\cdot(T_{at} - T_a)$, a lower – of heat loss in the channel walls $\alpha_{cfb}\cdot(T_f - T_b)$, which is described by the corresponding balance equation:

$$\alpha_{cpf}\cdot\left(T_p - T_f\right) = G_m \cdot c_p \cdot\left(T_p - T_g\right) + \alpha_{cfb}\cdot\left(T_f - T_b\right). \tag{6}$$

where α_{cpf} – heat transfer coefficient from the absorption panel to the coolant;
T_p – temperature of the absorption panel;
T_f – coolant temperature in the channel;
G_m – mass flow rate of coolant;
c_p – specific heat of coolant at constant pressure.

- Two heat loss flows from the panel $\alpha_{rpb}\cdot(T_p - T_b)$ and from the coolant $\alpha_{cfb}\cdot(T_f - T_b)$ are combined into losses through the header housing, which at $T_{ba} \approx T_a$ are estimated as $(\lambda/\delta)\cdot(T_b - T_a)$, and the balance on the inner wall of the coolant channel is reduced to the corresponding equation:

$$\alpha_{rpb}\cdot\left(T_p - T_b\right) + \alpha_{cfb}\cdot\left(T_f - T_b\right) = \left(\lambda/\delta\right)\cdot\left(T_b - T_a\right), \tag{7}$$

where λ – coefficient of thermal conductivity of the wall material;
δ – body wall thickness.

Moreover,

$$T_p = \left(\eta_0 \cdot E + \left(\alpha_{cpg} + \alpha_{rpg}\right)\cdot T_g + \alpha_{rpb}\cdot T_b + \alpha_{cpf}\cdot T_f\right) \Big/ \left(\alpha_{cpg} + \alpha_{rpg} + \alpha_{rpb} + \alpha_{cpf}\right). \tag{8}$$

All values of the composite equations depend on the input parameters of the process and can only be determined numerically by the method of sequential approximations (iterations), and the number of steps decreases when choosing initial values close to the initial values according to the literature [1]. Classical equations of liquid solar collectors and generalized relations for heat loss through glass are unsuitable for air collectors due to the complex dependence of the coefficient of total heat loss U_L from the design and flow modes; therefore, to assess the heat transfer coefficients, proven specialized methods are used without any doubt in the literature [2].

The total heat transfer coefficient to the inner surface of the glass is equal to:

$$\alpha_{pg} = \alpha_{cpg} + \alpha_{rpg} = N_u \cdot\left(\lambda/h_{pg}\right) + \varepsilon_{pg}\cdot\sigma\cdot\left(T_p^2 + T_g^2\right)\cdot\left(T_p + T_g\right), \tag{9}$$

where λ – air thermal conductivity coefficient;
N_u – Nusselt criteria;
h_{pg} – hydraulic channel height;
ε_{pg} – degree of blackness (effective emissivity) characterizing the radiant heat exchange between the absorption panel (p) and glass (g);
σ – radiation heat transfer coefficient.

The methodology for calculating the thermal parameters of the air collector should be based on numerical modeling using iterative methods, since the values are interrelated and depend on flow modes and design features; classical formulas of liquid collectors and generalized estimates of heat loss are unsuitable, therefore, proven specialized methods are used to accurately determine the heat transfer coefficients. The main tool for such analysis is the CFD (Computational Fluid Dynamics) method in SolidWorks, which allows to simulate and study thermal processes, hydro- and gas dynamics of flows in an integrated 3D CAD environment.

Results and discussion

The main method for performing heat transfer analysis by means of computer modeling is the CFD (Computational Fluid Dynamics) method in SolidWorks – a numerical method for modeling and analyzing liquid and gas flows, which is implemented in the SolidWorks Flow Simulation module. This tool is integrated directly into the 3D CAD SolidWorks environment and allows the engineer to conduct a detailed analysis of hydro- and gas-dynamics, thermal processes and the interaction of flows with solids. SolidWorks and SolidWorks Flow Simulation CFD software provide a complete engineering environment for heat transfer assessment, combining all phases of analysis in one package: from solid-state product modeling to setting heat transfer tasks, solving them, visualizing the results, optimizing the project and reporting. With SolidWorks Flow Simulation, you can focus on a detailed analysis of the temperature distribution in the liquid and solid parts of your product. This includes the ability to analyze complex physical processes such as thermal conductivity, heat convection, conjugate heat transfer between liquids and gases surrounding solid materials, radiation, heating, etc., using what-if scenarios, and then the engineer can quickly modify and optimize the design geometry in a 3D CAD tool. SolidWorks Flow Simulation works with all three heat transfer modes – thermal conductivity, convection and radiation, so it can analyze a wide range of applications. The starting point of any heat transfer analysis is to determine the general boundary conditions of the problem.

It is important to construct a computational grid, or cell, to display the complex geometry of the system during heat transfer analysis. SolidWorks Flow Simulation provides extensive visualization of what is happening with the heat dissipation of the structure, providing the engineer with valuable information that can help make design decisions. Visualization capabilities allow to more thoroughly explore the design. One way to study the temperature field is to use a section diagram that depicts the distribution of heat on a plane. The result chart can be displayed with any result parameter, and the view can be created as a contour chart, isolines, or vectors. In addition to sections, you can easily display a surface graph for any particular face, as well as automatically for the entire model.

Solving heat distribution problems is an iterative process. The software also assists in parametric analysis, for example, performing analysis repeatedly, for example, with different vent sizes to determine the optimal heat distribution design. Thus, SolidWorks Flow Simulation accelerates the iterative design process, allowing the engineer to quickly and easily implement the knowledge gained from the analysis into an improved project. SolidWorks Flow Simulation identifies hot spots, quantifies thermal efficiency, and ensures uniform heat distribution for heat transfer devices. In addition, SolidWorks Flow Simulation works simultaneously with the product development process and without the hassle of “dirty geometry” and grid generation using traditional CFD tools.

A solid three-dimensional calculation model of the air collector was created at the first stage of the research in the environment of the CAD module of the SolidWorks automated design system (Fig. 2). The design model consists of a wooden frame, glass walls and an absorbent element, which is a copper plate measuring 1000×2000×1 mm. The fan installation hole is provided for forced air extraction from the collector in its upper part. A number of holes are made for air inflow into the absorbing part of the structure in the lower end part of the collector.

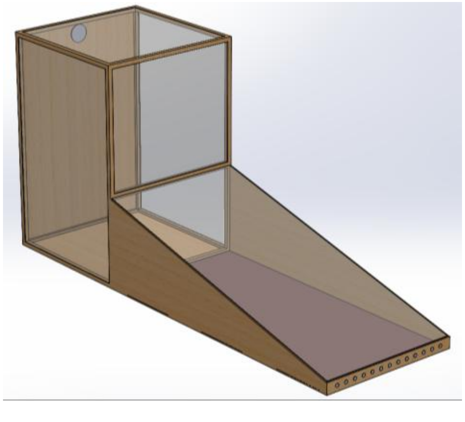


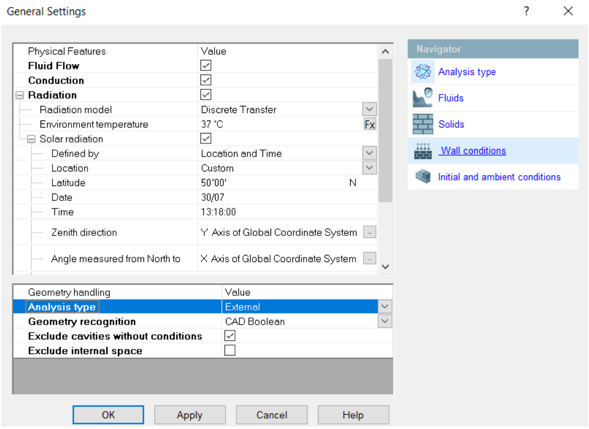
Fig. 2. **Design model of the air collector**

The process of studying the air collector consisted of two stages. Heating of the absorbing element (copper plate) was carried out at the first stage of process modeling. The second stage of modeling was based on the results of the first stage and consisted in the study of thermal processes in the reservoir space and visualization of the results obtained.

The type of analysis specified during the first stage of the absorber heating simulation is the external problem Flow Simulation, the ambient temperature, latitude – 50°00′, date – July 30, time – 1:18 p.m. are set in the study project as an input parameter (Fig. 3).

The collector grid model has been created. The study area is large enough to avoid the influence of boundary conditions on the results (Fig. 4).

The absorber material properties required for the calculation were imported from the plate model of the SolidWorks CAD module. The average value of the heat transfer coefficient and the average value of the plate surface temperature were selected as research objectives.


General Settings

Physical Features	Value
Fluid Flow	☑
Conduction	☑
Radiation	☑
Radiation model	Discrete Transfer
Environment temperature	37 °C
Solar radiation	☑
Defined by	Location and Time
Location	Custom
Latitude	50°00' N
Date	30/07
Time	13:18:00
Zenith direction	Y Axis of Global Coordinate System
Angle measured from North to	X Axis of Global Coordinate System

Geometry handling	Value
Analysis type	External
Geometry recognition	CAD Boolean
Exclude cavities without conditions	☑
Exclude internal space	☐

Fig. 3. **Setting the input modeling parameters**

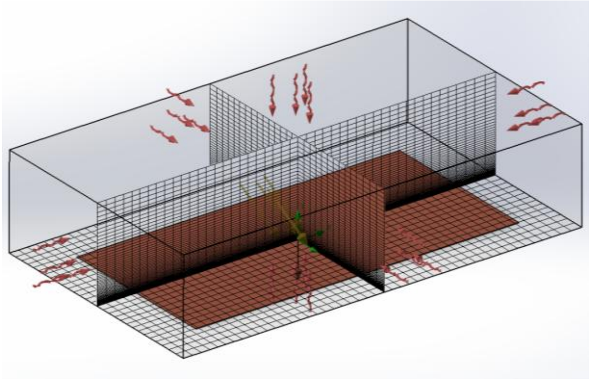


Fig. 4. **Grid model and study area**

The average value of the heat transfer coefficient, which is 4.45 W·(m^{-2}·K^{-1}), was established according to the results of the study (Fig. 5). The temperature distribution pattern over the surface of the absorbent plate was constructed (Fig. 6). The average surface temperature of the plate is 106.2 ºC.

The second stage of the research in SolidWorks Flow Simulation provided for the solution of an internal problem, the input parameters of which were the absorber temperature (106.2 ºC), as well as the boundary conditions at the inlet and outlet of the collector. Setting up the CFD analysis consisted in setting the parameters of the study, namely: working environment – air; ambient temperature – + 37ºC; acceleration of free fall (along the axis Y) – 9.81 m·s^{-2}; ambient pressure – 101325 Pa.

The size of the study area corresponds to the dimensions of the reservoir. Boundary conditions of the study (Fig. 5): mass flow rate of air at the fan outlet – 0.0588 kg·s^{-1}; pressure of the medium at the inlet holes of the collector model – 101325 Pa; ambient working temperature – +37 ºC; heat source – copper plate, temperature – +106.2 ºC.

The level of detail of the grid model is set – 4 with a gap size of 0.026 m, which ensures sufficient accuracy of the results for a relatively short duration of the modeling process.

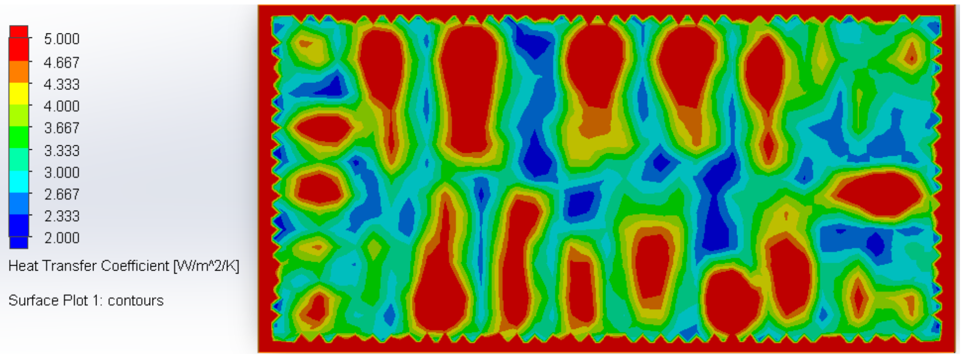


Fig.5. **Heat transfer coefficient distribution pattern on absorber surface**

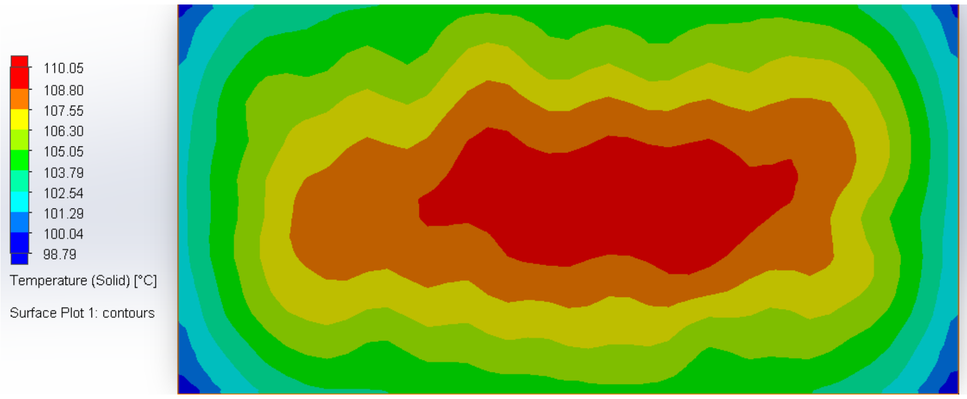


Fig. 6. **Temperature distribution pattern on absorber surface**

The second stage of the research in SolidWorks Flow Simulation provided for the solution of an internal problem, the input parameters of which were the absorber temperature (106.2 ºC), as well as the boundary conditions at the inlet and outlet of the collector. At the inlet – air temperature and ambient pressure, at the outlet – the mass flow rate of air by the fan, which is 0.05886 kg·s^{-1} (Fig. 7).

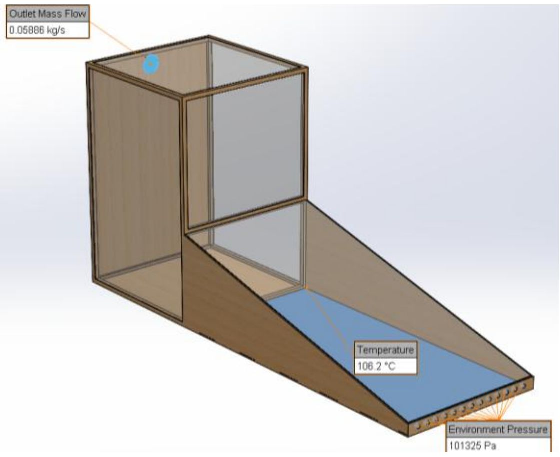


Fig. 7. **Setting the boundary conditions for the air collector study**

The picture of the air temperature distribution in the internal space of the collector, taking into account the heating of the absorber, the fan flow rate, the environmental parameters, as well as the design features of the air collector, was constructed based on the results of the study (Fig. 8).

In addition, the process of air movement and air mass current lines is visualized (Fig. 9). This makes it possible to visually assess the level of turbulence of air flows.

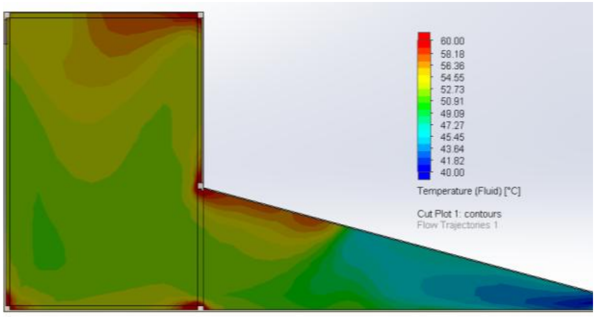


Fig. 8. **Air temperature distribution pattern**

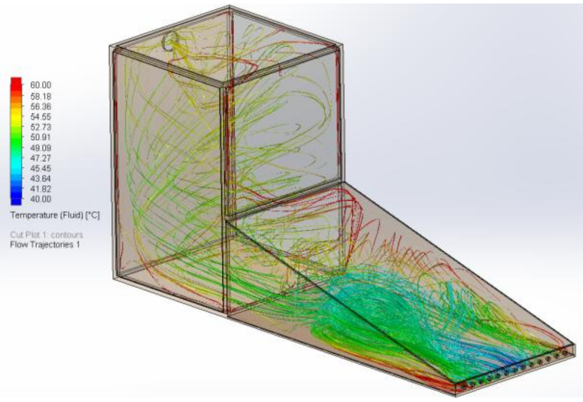


Fig. 9. **Visualization of the movement of air masses**

Consequently, the CFD method in SolidWorks Flow Simulation provides an efficient and integrated technique for numerical simulation of heat transfer, allowing the analysis of thermal conductivity, convection and radiation in complex structures simultaneously. This software environment combines modeling, setting boundary conditions, iterative solution, visualization of results and optimization of the project, which significantly speeds up the development process, allows to identify hot spots, evaluate the efficiency of heat transfer and make informed engineering decisions.

Conclusions

The method of calculating the thermal parameters of the air collector is effectively implemented through numerical modeling using iterative methods. All quantities are interrelated and depend on flow modes and design features. Classical formulas of collectors and generalized estimates of heat loss for air systems of solar dryers complicate the calculation methodology, therefore, specialized techniques are used to determine heat transfer coefficients accurately. The main tool for this analysis is CFD in the SolidWorks environment. This allows to model thermal, hydro- and gas-dynamic processes in an integrated 3D CAD environment. The results of the CFD analysis in SolidWorks Flow Simulation are in good agreement with the analytical data and complement them, while providing the opportunity to take into account various thermophysical parameters of the environment, namely, to change the conditions of air movement, the design of the collector and the fan flow rate, which provides an objective assessment of the efficiency of the collector and coolant.

The results obtained by CFD analysis in the SolidWorks Flow Simulation environment correspond to the results obtained by the analytical method and complement them. The advantage of using computer modeling of thermal processes in an air collector using SolidWorks Flow Simulation is the ability to set the values of various thermophysical parameters of the environment, regardless of the time of year, time and place of the computer experiment.

The next stage of work involves conducting full-scale tests of the air collector. This will allow checking the accuracy of numerical modeling and assessing the correspondence of the obtained theoretical results to real operating conditions. Such experiments will provide an opportunity to compare the predicted parameters of heat productivity and efficiency with actual indicators, determine the influence of external factors (ambient temperature, air flow rate) on the operation of the system, as well as clarify the design and operational recommendations for optimizing the operation of the collector in different modes.

Author contributions

Conceptualization, S.K., M.T., I.S., V.S. and N.T.; methodology, S.K. M.T., I.S. and N.T.; software, M.T., I.S. and N.T.; investigation, S.K., M.T., I.S. and V.S.; writing – original draft preparation, S.K. and I.S.; writing – review and editing, S.K., M.T., I.S., V.S. and I.K.; visualization, M.T., I.K. and N.T. All authors have read and agreed to the published version of the manuscript.

References

[1] Korobka S., Tolstushko M., Stukalets I., Sheichenko V., Tolstushko N. Study of energy-efficient characteristics of transparent enclosing constructions of solar dryers based on glass units. Engineering for rural development. Jelgava, 21.-23.05.2025. pp. 1019-1026. DOI:10.22616/ERDev.2025.24.TF213

[2] Pathare P. B. Effective moisture diffusivity of onion slices undergoing infrared convective drying. Biosystems Engineering. N. 93, 2006. pp. 285-291. DOI: 10.1016/j.biosystemseng.2005.12.010

[3] Goyal R. K. Kingsly A. R. P., Manikantan M. R., Ilyas S. M. Mathematical modeling of thin layer drying kinetics of plum in a tunnel dryer. Journal of Food Engineering. N. 79, 2007, pp. 176-180. DOI: 10.1016/j.jfoodeng.2006.01.041

[4] Garau M. C. Drying of orange skin: drying kinetics modeling and functional properties. Journal of Food Engineering. N. 7, 2006, pp. 288-295. DOI: 10.1016/j.jfoodeng.2005.04.017

[5] Menges H. O. Thin layer drying model for treated and untreated Stanley plums. Energy Conversion and Management. N. 47, 2006 pp. 23372348. DOI: 10.1016/j.enconman.2005.11.016

[6] Khazimov Z. M., Bora G. C., Khazimov K. M., Khazimov M. Z. Modeling of the motion of free convective drying agent in plastic helio dryer. Journal of Engineering Thermophysics, 23.4, 2014, pp. 306-315.
[7] Movassag S.Z., Zamzamian K. Numerical investigation on the thermal performance of double glazing air flow window with integrated blinds. Renewable Energy, 148, 2020, pp. 852-863, DOI: 10.1016/j.renene.2019.10.170
[8] Arici M., Kan M. An investigation of flow and conjugate heat transfer in multiple pane windows with respect to gap width, emissivity and gas filling. Renewable Energy, 75, 2015, pp. 249-256, DOI: 10.1016/j.renene.2014.10.004
[9] Nourozi B., Ploskić A., Chen Y., Ning-Wei Chiu J., Wang Q. Heat transfer model for energy-active windows - An evaluation of efficient reuse of waste heat in buildings. Renewable Energy, 162, 2020, pp. 2318-2329, https://doi.org/10.1016/j.renene.2020.10.043
[10] Kryvovyazyuk I.V., Strilchuk R.M. Strategic opportunities management at engineering enterprises. Actual Problems of Economics, vol. 183(9), 2016, pp. 144-155.