



Original Research Article

Computational Fluid Dynamics Investigation of Packed Bed Solar Air Heater by Aspect Ratio Optimization

Aya Ali Turki Jawad*, Mahmoud Mustafa Mahdi, Mohammed Jawad Mohammed

College of Electromechanical Engineering, University of Technology, Baghdad, Iraq

e-mail: Mohammed.j.mohammed@uotechnology.edu.iq, eme.23.109@grad.uotechnology.edu.iq,
50295@uotechnology.edu.iq

Cite as: Jawad, A. A. T., Mahdi, M. M., Jawad Mohammed, M., Computational Fluid Dynamics Investigation of Packed Bed Solar Air Heater by Aspect Ratio Optimization, J.sustain. dev. energy water environ. syst., 15(1), 1140776, 2027, DOI: <https://doi.org/10.13044/j.sdewes.d14.0776>

ABSTRACT

Packed-bed solar air heaters have attracted considerable interest due to their ability to improve heat transfer efficiency. This study numerically investigates the influence of channel aspect ratio on the thermo-hydraulic performance of a packed-bed solar air heater filled with mixed-size aluminium spherical elements. Three-dimensional CFD simulations were performed in ANSYS 2023 R1 under steady-state turbulent flow conditions using the standard $k-\epsilon$ model. Four channel heights (8, 10, 12, and 14 cm) were evaluated while maintaining identical operating conditions. The results show that channel height significantly affects both heat transfer and pressure loss. The 12 cm channel height provided the best overall performance, achieving the highest outlet air temperature and thermal efficiency with a relatively low pressure drop. Compared with the lowest aspect ratio, the optimum configuration increased thermal efficiency by approximately 85% and the thermo-hydraulic performance factor by about 76%. These findings provide practical guidance for the design and optimization of high-performance packed-bed solar air heaters.

KEYWORDS

Aluminum balls, Aspect ratio Optimization, Computational Fluid Dynamics, Solar air heater, Thermo-hydraulic factor, Mixed-size spherical packing, Thermal efficiency.

INTRODUCTION

Solar air heaters represent a viable and sustainable technology for low and medium-temperature thermal applications, including space heating, drying, and industrial preheating. Nevertheless, the thermal performance of conventional flat plate SAHs remains inherently limited due to weak convective heat transfer and the formation of stable thermal boundary layers along the absorber surface. To mitigate these limitations, numerous passive heat transfer enhancement techniques have been proposed, such as Various passive heat transfer enhancement techniques have been proposed, particularly packed-bed configurations. Among these approaches, packed beds composed of spherical elements have demonstrated considerable potential. Packed beds employing mixed-size spherical elements enhance turbulence, improve air mixing, and increase the effective heat transfer area, resulting in better thermo-hydraulic performance than smooth ducts. further complicates the internal flow structure, promoting enhanced mixing and improved thermo-hydraulic behaviour compared to smooth-duct systems. Despite these improvements, channel aspect ratio remains one of the

* Corresponding author

most influential geometric parameters affecting thermo-hydraulic performance. The packed bed SAHs are strongly governed by geometric parameters, particularly the channel aspect ratio, defined as the ratio of channel width to height. This parameter significantly influences velocity distribution, turbulence characteristics, residence time, pressure drop, and ultimately the balance between thermal enhancement and hydraulic losses. Although several studies have examined geometric modifications addressing the optimization of aspect ratio in multi-size packed-bed configurations remain limited.

Accordingly, the present study conducts a three-dimensional steady-state computational fluid dynamics analysis to evaluate the influence of channel aspect ratio on the thermo-hydraulic performance of packed-bed solar air heater under constant width conditions, to identify the optimal geometric configuration that maximizes heat transfer enhancement while maintaining acceptable hydraulic performance. Abdulmajeed *et al.* [1] numerically compared flat, grooved, and finned parallel-pass solar air heaters, showing that geometry strongly governs heat transfer, efficiency, and pressure losses. Nguyen Van Hap *et al.* [2] integrated latent heat storage and fins in solar air heaters, improving efficiency, stabilizing outlet temperature, and extending operating time. Chamarthi and Singh [3] reviewed experimental methods and enhancement techniques for solar air heaters, identifying optimal flow, geometry, and turbulence strategies improving efficiency. An *et al.* [4] numerically studied finned solar air heaters with sequential obstacles, revealing that optimal geometry improves heat transfer and thermo-hydraulic performance. Ghanem and Bhosale [5] experimentally and numerically optimized honeycomb roughness solar air heaters, improving thermal exergy efficiency, reducing emissions, and enhancing economic feasibility. Khatri *et al.* [6] experimentally improved solar air heater performance using an arched wavy absorber and porous fins, achieving higher efficiency and optimal airflow. Arunkumar *et al.* [7] reviewed solar air heater enhancement methods, concluding that artificial roughness, optimized geometry, and flow control significantly improve thermo-hydraulic efficiency performance. Sornek and Papis-Frączek experimentally and numerically developed a solar air heater with thermal storage, demonstrating improved ventilation preheating, efficiency, and building energy savings. Salih *et al.* [8] numerically and experimentally compared solar air heaters with PCM, showing that solar flux and airflow strongly influence temperature rise and effectiveness.

Uzakov *et al.* [9] experimentally investigated a solar air heater with an accumulator heat exchanger, achieving stable outlet temperatures, enhanced heat storage, and improved daily efficiency. Kazemi Moghadam *et al.* [10] numerically analyzed an arc-rib solar air heater, finding that optimized pitch and aspect ratio significantly enhance heat transfer performance. Siddique *et al.* [11] experimentally optimized ribbed and corrugated solar air heater channels, finding that 45° corrugation yields the best thermal performance and efficiency. Nashee and Mushatet [12] experimentally and numerically evaluated ribbed rectangular ducts, finding intermittent ribs maximized heat transfer and overall performance despite higher friction, with close agreement between CFD and experimental results. Al-Chlahawi *et al.* [13] numerically analyzed trapezoidal-rib solar air heater ducts, showing that artificial roughness enhances heat transfer, turbulence, and thermo-hydraulic performance significantly. Addini and Nassab [14] numerically investigated a V-corrugated grooved solar air heater, showing that suction-jet interaction and geometry optimization significantly enhance efficiency and performance. Ali *et al.* [15] optimized dimpled blended-wing-body aerodynamics using computational fluid dynamics and Taguchi, identifying diameter dominance and optimal configurations improving drag reduction and lift-to-drag ratio. Shaik *et al.* [16] numerically evaluated triangular-duct solar air heaters with varied rib roughness, revealing that semicircular ribs maximize thermo-hydraulic performance and heat transfer.

Ferreira and Silva [17] developed a validated costing model for residential solar thermal systems, linking collector area and storage size with investment costs. Bowley and Evins [18] optimized integrated building energy systems using energy-hub modeling, demonstrating that renewable integration reduces costs, emissions, and improves economic performance. Singh

et al. [19] reviewed solar air heaters, highlighting roughness, flow design, and optimization strategies to enhance heat transfer and overall thermo-hydraulic efficiency. Murali *et al.* [20] experimentally assessed double-pass finned solar air heaters, finding that lower-channel fins significantly improved heat transfer, efficiency, and overall thermal performance. Boulemtafes-Boukadoum *et al.* [21] numerically showed that aspect ratio and buoyancy govern mixed convection, revealing optimal channel height enhancing heat transfer in solar air heaters. Menasria *et al.* [22] ANSYS Fluent and the RNG $k-\varepsilon$ turbulence model were used to numerically simulate a flat plate solar air heater duct with continuous rectangular baffles. The performance reached its peak with $THPF$ of 0.857 at $BR=0.7\sigma$, $PR=2$, and $Re=5000$. Heat transfer enhancement, turbulence intensity, and recirculation regions significantly improved the duct's thermal behaviour. Ahmed *et al.* [23] performed a numerical investigation of a solar air heater with spherical ribs via ANSYS Fluent and the $k-\varepsilon$ turbulence model. The 5000–15000 range of Reynolds numbers was given for the less significant heat fluxes (under 1000 W/m²). The spherical rib shape significantly improved the heat transport, Nusselt number and turbulence intensity with maximum thermo-hydraulic performance factor ($THPF$) of 2.3 under low Reynolds number conditions. Sharma *et al.* [24] numerical study of a solar air heater with roughness in the form of protruded square shape was investigated by using ANSYS Fluent at Reynolds numbers from 4000 to 20000 under the RNG $k-\varepsilon$ model. The best performance was obtained at $p/e = 38.8$ and $e/D = 0.036$ with thermal efficiency equal to 63.17%, along with significant enhancement in Nusselt number, turbulence intensity, and heat transfer attributes.

Although previous studies have investigated surface roughness, fins, baffles, and other single geometric modifications to improve solar air heater performance, the combined influence of mixed-size aluminium spherical packing and channel aspect ratio has received limited attention. Therefore, the present study numerically investigates the thermo-hydraulic performance of a packed-bed solar air heater by varying the channel height while maintaining a constant channel width. The novelty of this work lies in identifying the optimum channel aspect ratio that maximizes heat transfer enhancement while maintaining acceptable hydraulic losses through three-dimensional CFD simulations.

MATERIALS AND METHODS

This section presents the numerical methodology, physical model, governing equations, boundary conditions, and mesh independence analysis used to evaluate the thermo-hydraulic performance of the packed-bed solar air heater under different channel aspect ratios.

Geometry and physical model

A three-dimensional numerical simulation of a solar air heater was performed using ANSYS 2023 R1 to investigate the effect of air channel height on thermal performance and heat transfer. The solar air heater consists of a single design with fixed dimensions (120 cm, 80 cm). The air channel contains aluminum balls of different diameters used as heat transfer enhancement elements inside the duct. Along the length of the channel, aluminium spherical elements with diameters of 2.8 cm, 4.5 cm, and 6 cm were arranged to improve airflow mixing via high turbulence intensity, increase heat transfer surface area, and minimize stagnant flow regions within the channel. The existence of the mixed-size arrangements facilitates a much more uniform thermal distribution in contrast to uniform spherical packing. Spherical aluminium elements with different diameters were selected to enhance turbulence intensity, increase the heat transfer surface area, and disrupt thermal boundary layers inside the air channel. The mixed-size arrangement improves airflow distribution and reduces stagnant flow regions compared with uniform spherical packing. Only the channel height was varied while all other parameters were kept constant. Four different channel heights were studied: (8 cm, 10 cm, 12 cm, 14 cm) as shown in [Figure 1](#). The inlet air temperature, air velocity, and applied heat flux were kept constant in all simulations in order to study only the effect of channel height

on the thermal performance of the solar air heater. Air was used as the working fluid and assumed to be incompressible with temperature-dependent properties. The flow inside the channel was considered steady, three-dimensional, and turbulent.

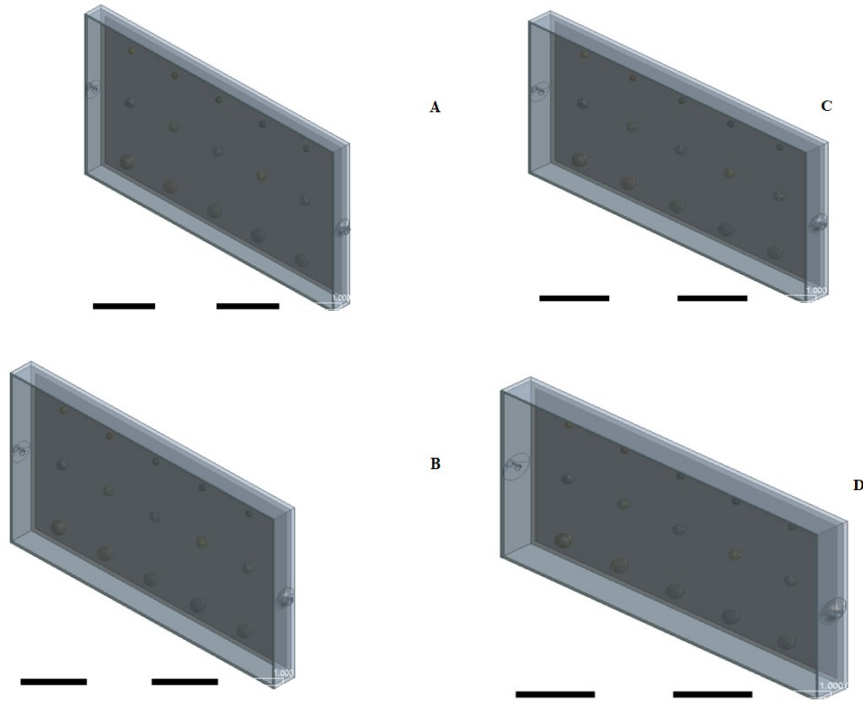


Figure 1. Geometry of the packed-bed solar air heater with different channel heights:
A) 8 cm; B) 10 cm; C) 12 cm; D) 14 cm

Governing equations

The airflow inside the SAH was modeled as a steady, three-dimensional, and turbulent flow. Air was considered an incompressible fluid with temperature-dependent thermophysical properties. The governing equations adopted in this study were based on the Reynolds-averaged Navier-Stokes (RANS) formulation for turbulent incompressible flow, as described by Tannehill *et al.* [25]. The governing equations are given as follows:

- **Continuity Equation**

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x} = 0 \quad (1)$$

- **Reynolds-Averaged Momentum Equation (RANS)**

$$\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] - \frac{\partial}{\partial x_j} (\rho \overline{u_i' u_j'}) \quad (2)$$

- **Energy Equation**

$$\rho C_p \left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} \right) = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \quad (3)$$

where $(\rho \overline{u_i' u_j'})$ is the Reynolds stress tensor, representing the additional momentum transport due to turbulent fluctuations. In the present study, the Reynolds stresses were modeled using the standard k-epsilon turbulence model based on the Boussinesq eddy-viscosity hypothesis.

Performance indicators

The following equations define the performance indicators used in this study.

Eqs. ((1–(4) were adopted based on the conventional heat transfer and thermo-hydraulic relations reported in the literature [26], [27].

1) Thermal Efficiency

$$\eta = \frac{Q_u}{I A} \quad (4)$$

where:

$$Q_u = m C_p (T_{\text{out}} - T_{\text{in}})$$

and:

- Q_u is the useful heat gain (W);
- M is the air mass flow rate (kg/s);
- C_p is the specific heat capacity of air (J/kgK);
- T_{out} is the outlet air temperature (°C);
- T_{in} is the inlet temperature (°C);
- I is the solar heat flux (W/m²);
- A is the absorber area (m²).

2) Reynolds Number

$$Re = \frac{\rho u D_h}{\mu} \quad (5)$$

where:

$$D_h = \frac{2HW}{H + W}$$

and:

- D_h is the hydraulic diameter (m);
- H is the channel height (m);
- W is the channel width (m);
- μ is the dynamic viscosity (kg/m.s).
- u is the inlet air velocity (m/s).
- ρ is the air density (kg/m³).

3) Nusselt Number

$$Nu = \frac{h D_h}{k} \quad (6)$$

where:

- Nu is the Nusselt number
- h is the convective heat transfer coefficient (W/m²·K).
- k is the thermal conductivity of air (W/m·K).

4) Thermo-Hydraulic Performance Factor (THPF)

$$THPF = \frac{\left(\frac{Nu}{Nu_0}\right)}{\left(\frac{f}{f_0}\right)^{\frac{1}{3}}} \quad (7)$$

$$\Delta p = p_{in} - p_{out}$$

$$f = \frac{2 \Delta p D_h}{\rho u^2 L}$$

where:

- p is the pressure drop (Pa);
- p_{in} is the inlet pressure (Pa);
- p_{out} is the outlet pressure (Pa);
- f is the friction factor;
- f_0 is the friction factor of the reference case;
- Nu_0 is the Nusselt number of the reference case.

Boundary conditions

The boundary conditions used in the present numerical simulations were defined to represent steady-state airflow inside the solar air heater channel. The simulations were performed under steady-state turbulent flow conditions using the standard k- turbulence model coupled with Rosseland radiation model. Air was considered the working fluid, while aluminium was selected as the material of the spherical packing elements because of its high thermal conductivity and good heat transfer characteristics. A velocity inlet condition with a specified velocity and temperature was applied at the inlet, and a pressure outlet condition was imposed at the exit. The outlet pressure was defined as zero gauge pressure, while gravity effects were included in the vertical direction. A uniform heat flux was applied to the absorber plate and spherical aluminium packing elements. The external walls were assumed to be adiabatic. The main boundary conditions used in the simulations are summarized in [Table 1](#).

Table 1. Boundary conditions used in the numerical simulation

Parameter	Value
Fluid	Air
Simulation type	Steady-state
Turbulence model	Standard k-ε
Inlet velocity	0.3-1.2 m/s
Radiation model	Rosseland
Sphere material	Aluminum
Inlet temperature	30 °C
Outlet pressure	0 Pa (gauge pressure)
Heat flux	200 - 800 W/m ²
Channel height	8 - 14 cm
Walls	Adiabatic
Gravity	Enabled

Mesh and independence test

The domain of the packed-bed solar air heater was discretized using ANSYS Meshing. The air channel was filled with a complex packed-bed geometry which consisted of circular aluminum balls, and therefore an unstructured mesh composed almost entirely of tetrahedral elements was used in order to characterize the detailed flow heat transfer within the air channel. A mesh refinement was performed for the heated absorber surface and in regions of spherical aluminum balls because large velocity and temperature gradients are predicted. The face and body sizing technique was applied to increase mesh resolution in these important regions. A total of around 1.24 million elements and 380762 nodes were used to create the final computational grid, which offered an appropriate balance between accuracy and simulation time.

The mesh quality parameters were kept within the range of acceptable computational fluid dynamics limits to provide stable numerical convergence and representative thermo-hydraulic predictions. Finally, a mesh independence test was performed to guarantee that numerical results were not sensitive to different mesh resolutions. Comparisons of all mesh configurations were made with respect to outlet air temperature and thermodynamic parameters.

The results obtained demonstrated negligible differences after the selected mesh size, showing stable and reliable numerical predictions. Thus, the adopted mesh setup was believed to be adequately fine for the current simulations. [Table 2](#) summarizes the mesh specifications. The overall computational mesh, local refinement around the spherical elements, and the cross-sectional grid are presented in figures Figure 2–Figure 5.

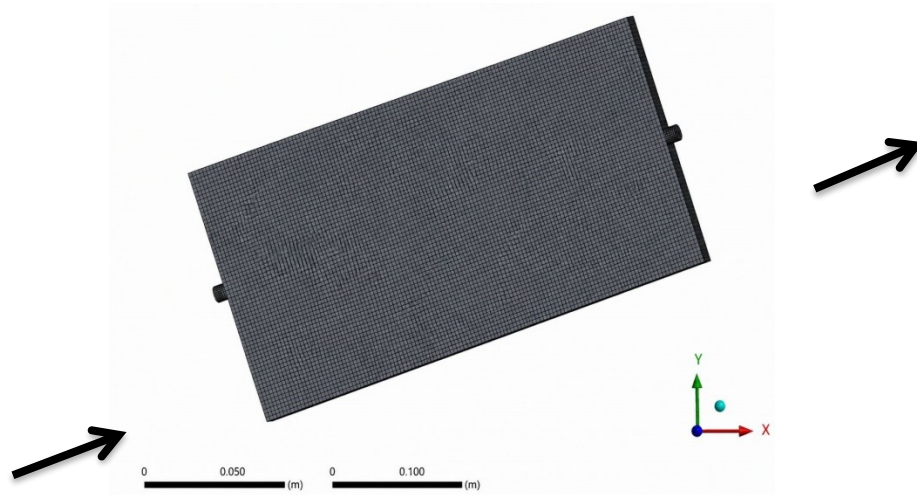


Figure 2. Overall computational mesh of the solar air heater model

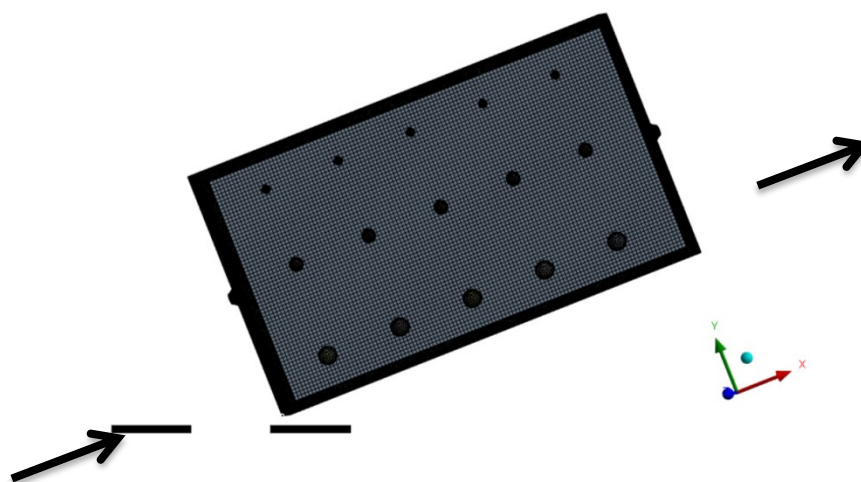


Figure 3. Numerical mesh generated for the SAH model

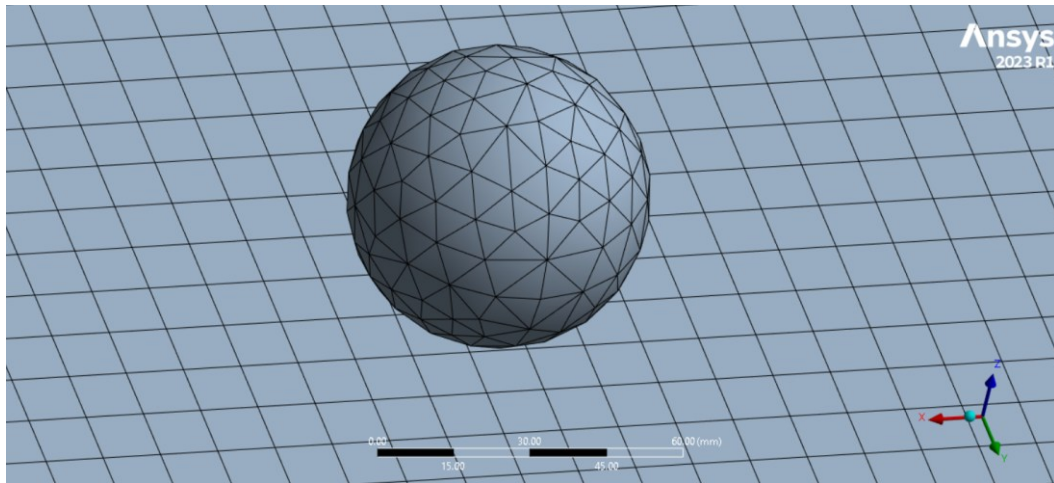


Figure 4. Local mesh refinement around the spherical aluminum balls inside the air channel

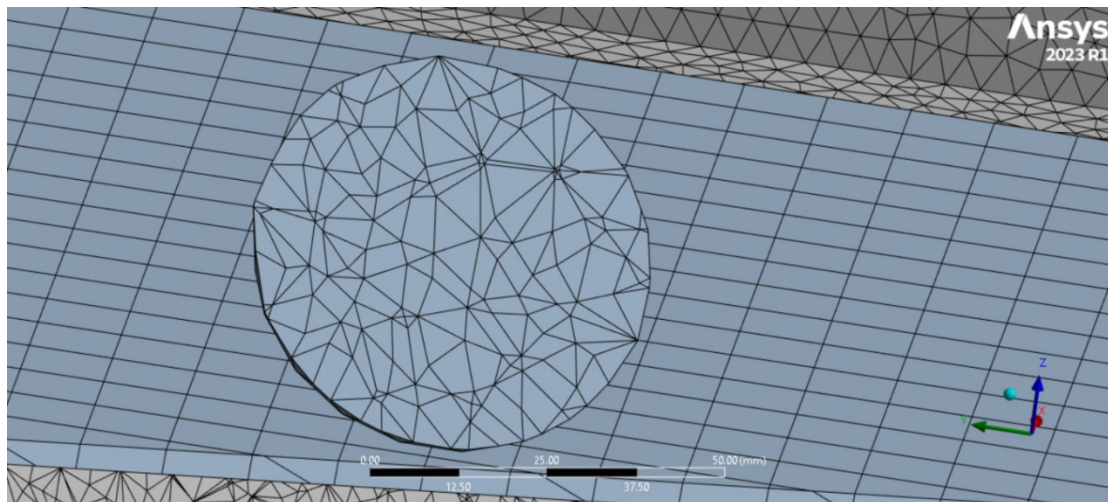


Figure 5. Cross-sectional view of the computational mesh inside the air channel

Table 2. Mesh specifications and quality parameters for the solar air heater model

Mesh Attribute	SAH
Element size	5 mm
Number of Nodes	380762
Number of Elements	1,266,459 M
Mesh Metric	
1-Arthogonal	Max=1
2-Skewness	Max=0.5
3-Aspect Ratio	Max=5
Growth Rate	1.2
Method	Face Sizing, Body Sizing, and Tetrahedral.

The percentage error between different mesh sizes was calculated using the following equation:

$$\text{Error(\%)} = \frac{X_i - X_{\text{ref}}}{X_{\text{ref}}} \times 100 \quad (8)$$

where X_i represents the calculated value for each mesh size, and X_{ref} is the reference value obtained from the finest mesh configuration. The mesh independence results for the packed-bed solar air heater at a channel height of 12 cm are presented in [Table 3](#). Five mesh resolutions were evaluated by comparing the temperature difference, pressure drop, and percentage error to identify a suitable mesh with acceptable accuracy and computational cost.

Table 3. Mesh independence test for the present SAH model (H=12 cm)

Elements size	Elements	Mesh	ΔT (°C)	Δp (Pa)	Error %
12 mm	106,960	Coarse	21.225	1.19	10.05 %
10 mm	171,143	Medium	21.259	1.24	10.23 %
8 mm	352,660	Fine	21.171	1.32	9.77 %
6 mm	689,925	Finer	19.968	1.34	3.53 %
5 mm	1,266,459	Finest	19.286	1.41	0 %

The variation of pressure drop with increasing number of mesh elements is plotted in [Figure 6](#). The pressure drop gradually increases as the mesh becomes finer, then stabilizes at higher mesh densities, indicating acceptable mesh convergence and numerical stability.

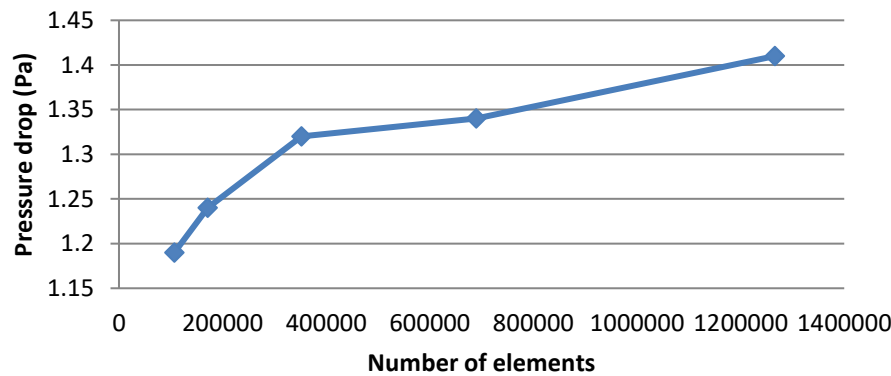


Figure 6. Mesh independence test for pressure drop

The effect of mesh density on the air temperature rise is illustrated in [Figure 7](#). The results demonstrate that the variation in temperature rise becomes smaller with finer meshes, confirming that the selected mesh provides reliable thermal predictions with acceptable accuracy.

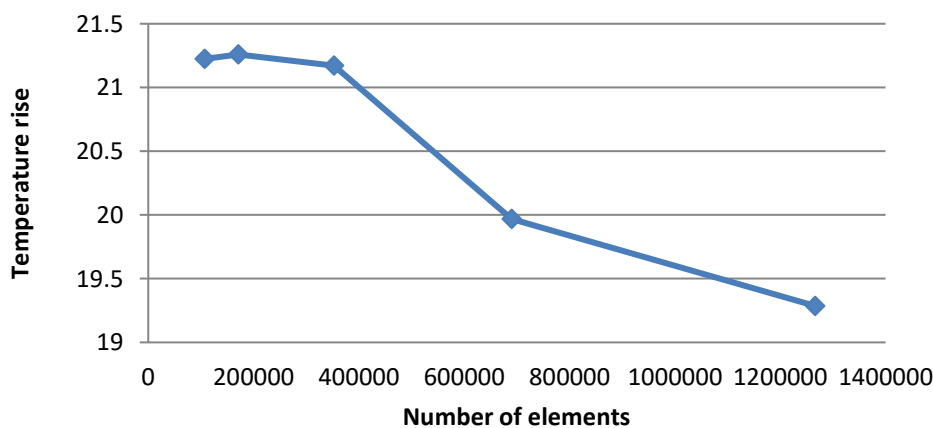


Figure 7. Mesh independence test for temperature rise

Based on the mesh independence study, the mesh consisting of approximately 1.266 million elements was selected for all subsequent simulations because further mesh refinement produced negligible changes in the predicted temperature rise and pressure drop. Accordingly, this mesh was considered sufficiently fine to provide mesh-independent thermo-hydraulic predictions while maintaining an acceptable computational cost.

Model validation

The numerical model was validated by performing a mesh independence test. The numerical simulation was performed under steady-state conditions. Several simulations were carried out using different mesh densities to ensure that the results were not affected by the number of elements. The comparison showed that the variation in outlet air temperature and pressure drop became negligible beyond a certain mesh size. Therefore, the mesh containing approximately 1,266,459 elements was selected, as it provides a suitable balance between computational accuracy and simulation time.

To further confirm the reliability of the developed computational fluid dynamics model, the predicted thermohydraulic behaviour was compared with the numerical study of Ebadi *et al.* [28], who investigated the performance of packed-bed solar air heaters using three-dimensional CFD simulations. Although the packed-bed configuration differs from the present study in terms of particle geometry and arrangement, both investigations exhibit similar thermo-hydraulic trends. In both studies, increasing the Reynolds number enhanced the heat transfer while simultaneously increasing the pressure drop due to stronger turbulence and higher flow resistance within the packed bed.

The agreement in these overall trends confirms that the present numerical model provides reliable predictions of the thermo-hydraulic performance of packed-bed solar air heaters. A comparison between the present results and those reported by Ebadi *et al.* is presented in Table 4.

Table 4. Comparison of the present numerical results with the computational fluid dynamics study of Ebadi *et al.* [28]

Parameter	Present study	Ebadi <i>et al.</i>
Numerical method	ANSYS 2023 R1	STAR-CCM+
Packed bed	Mixed-size aluminium sphere	Cylindrical packed bed
Flow regime	Turbulent	Turbulent
Heat transfer	Increased with Reynolds number	Increased with Reynolds number
Pressure drop	Increased with Reynolds number	Increased with Reynolds number
Overall trend	Good agreement	Reference
Maximum pressure drop	1.8 Pa	Higher than the present study due to a denser packed-bed configuration.
Heat transfer enhancement	85 % (at H= 12 cm)	25-35%

The convergence history of the scaled residuals for the 12 mm and 5 mm meshes used in the mesh independence study is illustrated in Figure 8. In both cases, all residuals (continuity, momentum, energy, and turbulence variables) decrease rapidly during the initial iterations and reach stable low levels within the accepted convergence limits. The coarser mesh (12 mm) exhibits slight oscillations due to flow disturbances around the packed elements, while the finer mesh (5 mm) demonstrates smoother and more stable convergence with lower residual magnitudes. This

behaviour confirms numerical stability and indicates that the selected fine mesh provides reliable and mesh-independent thermo-hydraulic predictions.

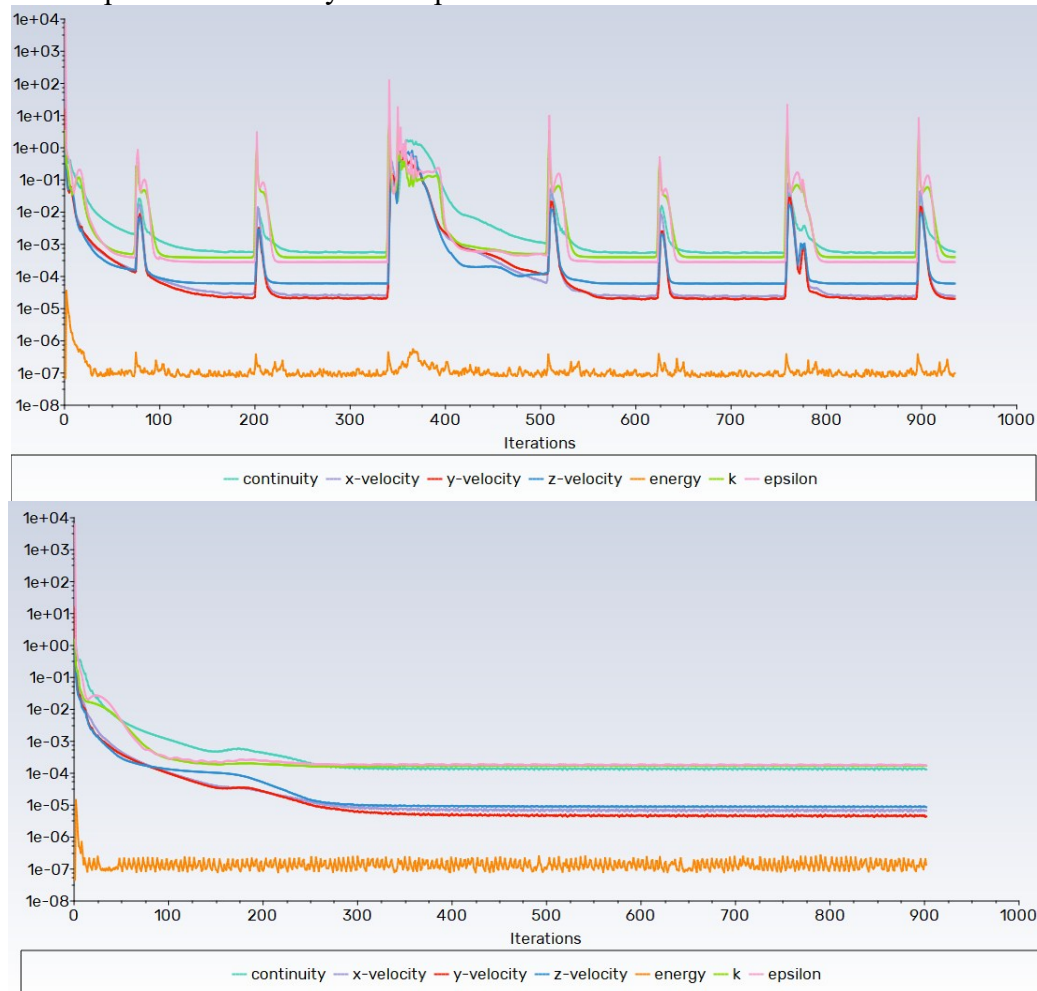


Figure 8. Convergence history of scaled residuals for coarse and fine meshes

RESULTS AND DISCUSSION

The Reynolds number calculated for different channel heights is presented in [Table 5](#) based on the hydraulic diameter of the air duct. The Reynolds number increases with channel height due to the enlargement of the hydraulic diameter. All values exceed 2.3×10^3 , confirming turbulent flow conditions within the SAH for all investigated cases.

Table 5. Reynolds number at different channel heights

Reynolds number (Re)	Hydraulic diameter Dh (m)	Channel heights (cm)
2.9×10^3	0.145	8
3.6×10^3	0.178	10
4.1×10^3	0.207	12
4.7×10^3	0.235	14

Flow pattern and temperature distribution

The airflow and temperature behavior inside the SAH were analyzed to evaluate the thermal performance of the packed-bed configuration. The temperature contour inside the solar air heater at a channel height of 12 cm ([Figure 9](#)). The air temperature gradually increases along the flow direction due to heat transfer from the heated absorber plate to the flowing air. The

presence of spherical aluminum balls enhances turbulence and improves mixing inside the channel, which leads to better convective heat transfer.

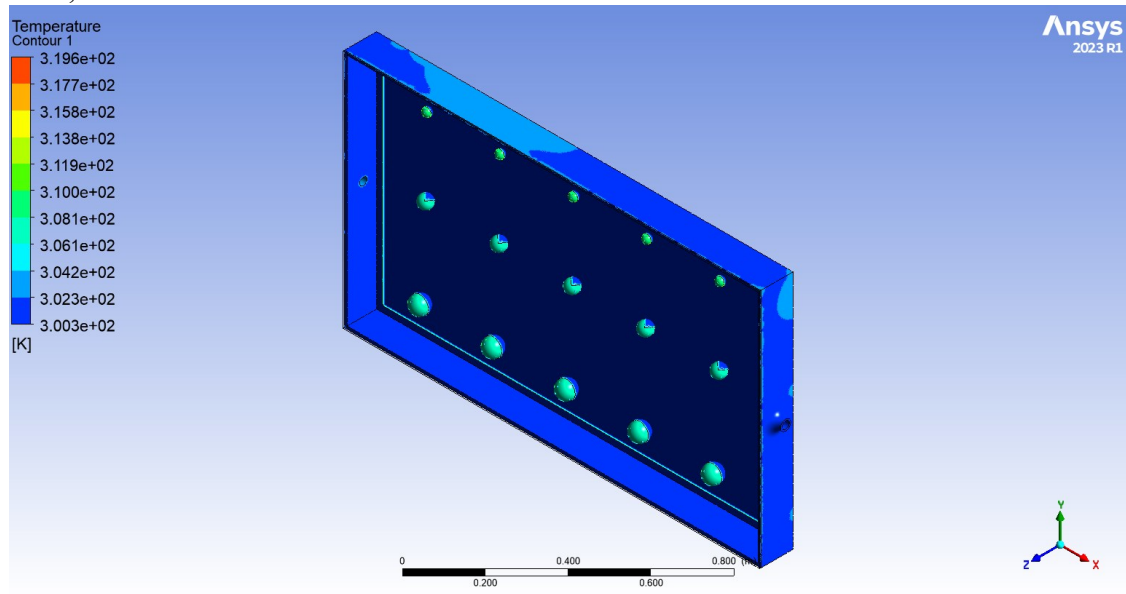


Figure 9. Temperature contour inside the SAH for channel 12 cm

Furthermore, the velocity streamlines presented in **Figure 10** illustrate the airflow pattern inside the channel. The streamlines clearly show that the air flows smoothly around the spherical elements, generating secondary flows and local turbulence. This behavior enhances the interaction between air and heated surfaces, resulting in improved thermal performance. The combined effect of temperature distribution and flow mixing confirms that the packed-bed configuration significantly enhances heat transfer within the solar air heater.

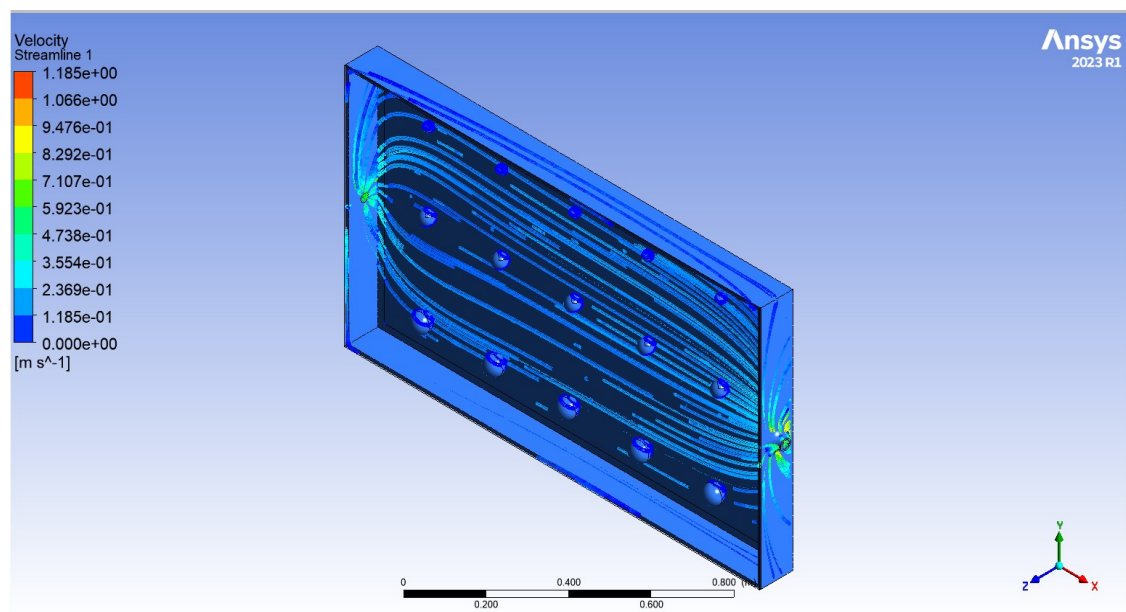


Figure 10. Velocity streamlines inside the SAH for the optimal channel height

Effect of aspect ratio

The effect of channel aspect ratio on the performance of the solar air heater was numerically investigated by varying the channel height while keeping all operating conditions constant. The inlet air temperature was fixed at 30 °C, the inlet velocity was maintained at 0.3 m/s, and a uniform heat flux of 800W/m² was applied to the absorber surface. Four channel heights (8,

10, 12, and 14 cm) were examined to evaluate their influence on thermal behavior and pressure loss. Changing the channel height significantly affects the airflow structure inside the heater. At lower channel heights, the airflow becomes more confined, which enhances mixing and improves convective heat transfer between the absorber surface and the air. However, this also increases flow resistance and results in higher pressure losses. Conversely, increasing the channel height reduces frictional resistance and pressure drop, but excessive height weakens the interaction between the airflow and the heated surfaces, leading to a slight reduction in heat transfer effectiveness. The numerical results indicate that the channel height of 12 cm provides the best balance between heat transfer enhancement and pressure loss. At this configuration, a higher temperature rise was achieved while maintaining a moderate pressure drop compared to other cases. Thus, the 12 cm channel height was identified as the optimal aspect ratio, offering the most favorable thermo-hydraulic performance for the investigated solar air heater, as shown in **Figure 11**.

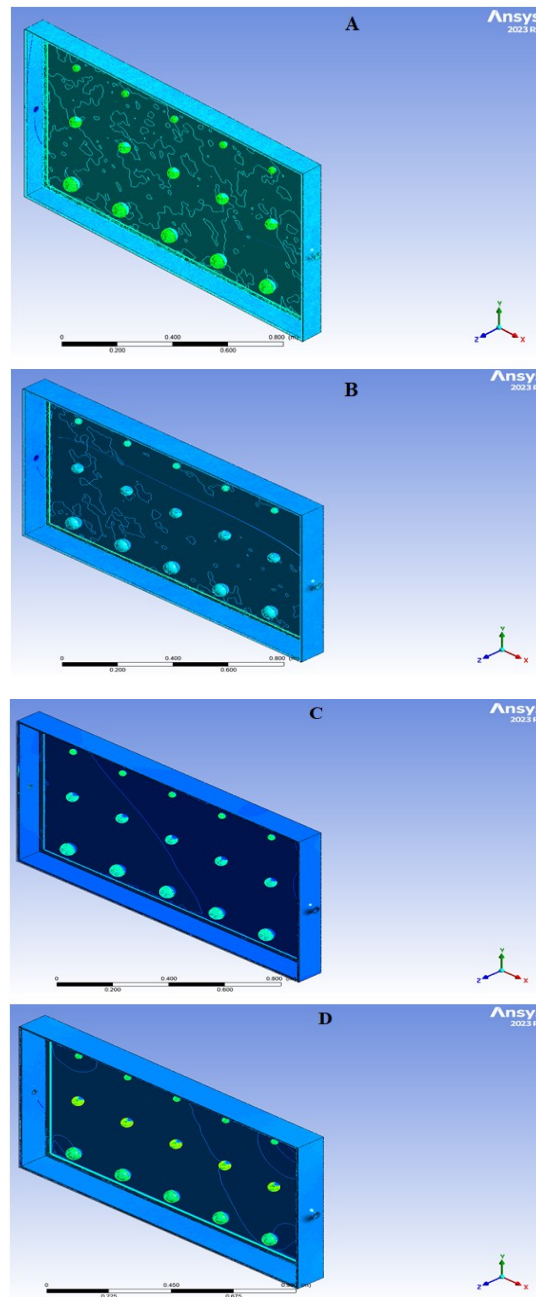


Figure 11. Variation of channel aspect ratio in the SAH model: A) 8 cm; B) 10 cm; C) 12 cm; D) 14 cm

Heat transfer performance

The heat transfer behavior of the solar air heater was evaluated for different channel heights, as summarized in [Table 6](#). The variation in channel height has a clear influence on the outlet air temperature and the temperature rise across the collector. At smaller channel heights, the airflow is strongly confined, which enhances contact between the air and the heated surfaces. This interaction improves energy absorption but is accompanied by higher resistance to flow. As the channel height increases, the air passage becomes wider, reducing confinement and slightly lowering the heat exchange intensity. The results indicate that the configuration with a channel height of 12 cm achieved the highest outlet temperature and the largest temperature rise. This configuration also provided the maximum thermal efficiency among all tested cases.

The improved performance at this height can be attributed to the balanced interaction between airflow mixing and available heat transfer area inside the duct. Although the 8 cm configuration produced a noticeable temperature increase, it experienced higher flow resistance. On the other hand, increasing the height to 14 cm reduced the pressure losses but slightly decreased the heat transfer effectiveness. Accordingly, the 12 cm configuration represents the most favorable condition for thermal energy gain under the present operating conditions. These findings confirm that channel height plays a critical role in determining the thermal response of the SAH and must be carefully optimized to achieve efficient performance. The Nusselt number increases with channel height due to the enlargement of the hydraulic diameter, which enhances convective heat transfer. Meanwhile, the friction factor decreases as the channel height increases, indicating lower flow resistance within the air duct.

Table 6. Thermal and thermo-hydraulic performance at different channel heights

Channel height (cm)	Outlet temperature (°C)	Temperature rise ΔT (°C)	Pressure drop (Pa)	Nu	f	Thermal efficiency (%)	Thermo-hydraulic performance
8	45.4	15.4	1.8	26.3	0.013	46 %	1
10	45.2	15.2	1.4	32.2	0.010	56 %	1.21
12	46.4	19.2	1.3	37.4	0.009	85 %	1.76
14	43	14.4	1.0	42.4	0.008	74 %	1.68

The influence of channel height on the temperature rise is presented in [Figure 12](#). The results reveal a clear improvement in thermal performance as the height increases up to 12 cm, where the maximum temperature rise is achieved. Beyond this value, further increase in height leads to a decline due to weakened convective interaction and reduced heat transfer effectiveness. Accordingly, a channel height of 12 cm represents the optimal configuration, providing the best thermo-hydraulic performance among the investigated cases.

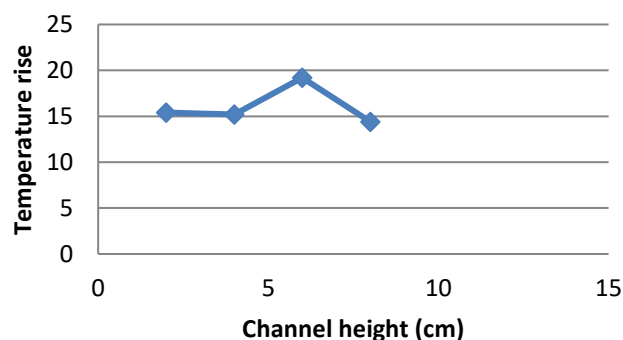


Figure 12. Temperature rise versus channel height

The variation of the Nusselt number with channel height is presented in **Figure 13**. The Nusselt number increases as the channel height increases due to the larger hydraulic diameter, which enhances convective heat transfer inside the solar air heater.

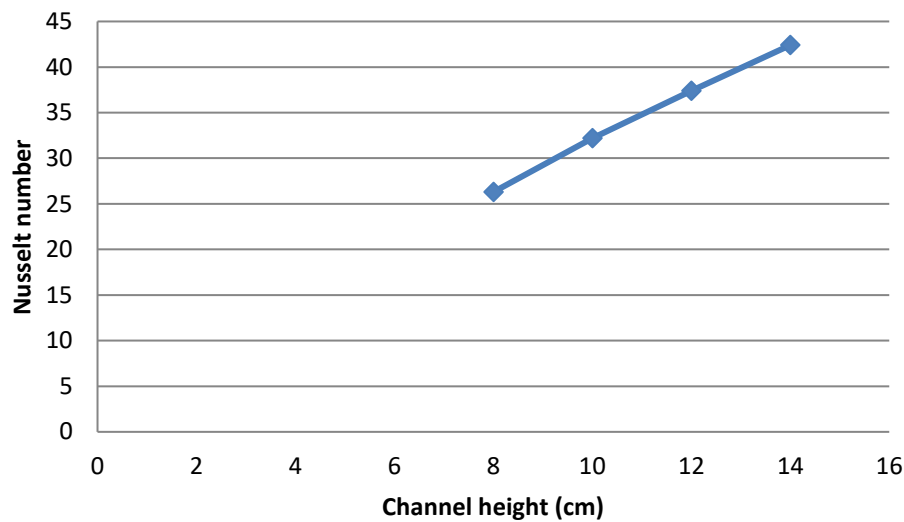


Figure 13. Variation of Nusselt number with channel height

Pressure drop

The pressure drop decreases progressively as the channel height increases from 8 cm to 14 cm. Smaller heights generate stronger flow confinement and higher frictional resistance, resulting in greater pressure losses. Conversely, larger channel heights reduce hydraulic resistance due to smoother airflow development. Although the 14 cm case provides the lowest pressure drop, the 12 cm configuration offers a better balance between thermal enhancement and acceptable hydraulic losses, as shown in **Figure 14**.

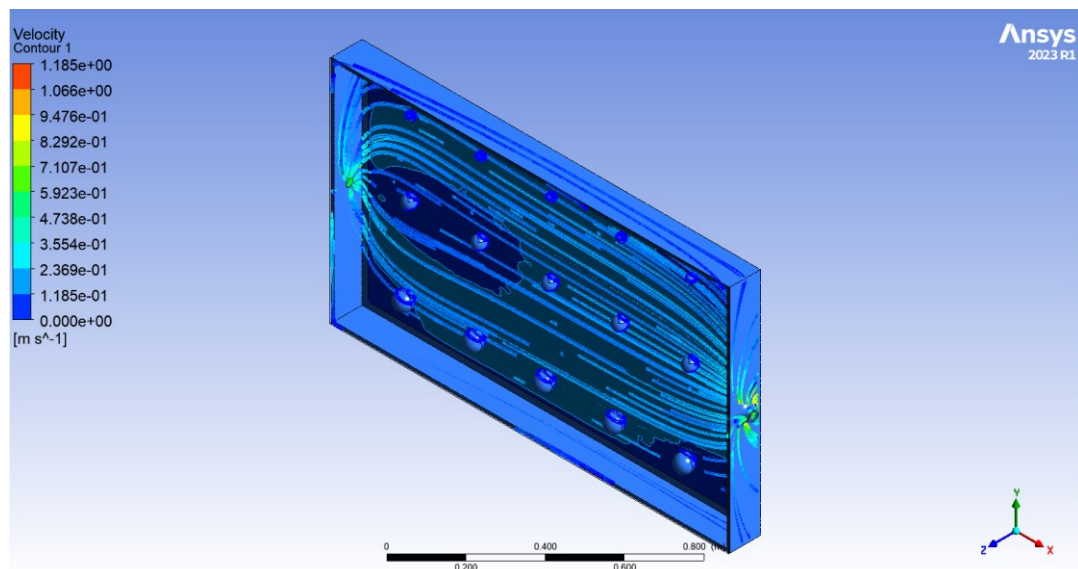


Figure 14. Velocity contours and streamlines inside the SAH for the optimal channel height (12 cm)

The pressure drop decreases as the channel height increases due to reduced flow resistance, is presented in **Figure 15**. While the lowest pressure drop occurs at 14 cm, the 12 cm channel height provides a better balance between thermal performance and pressure loss, making it the optimal configuration.

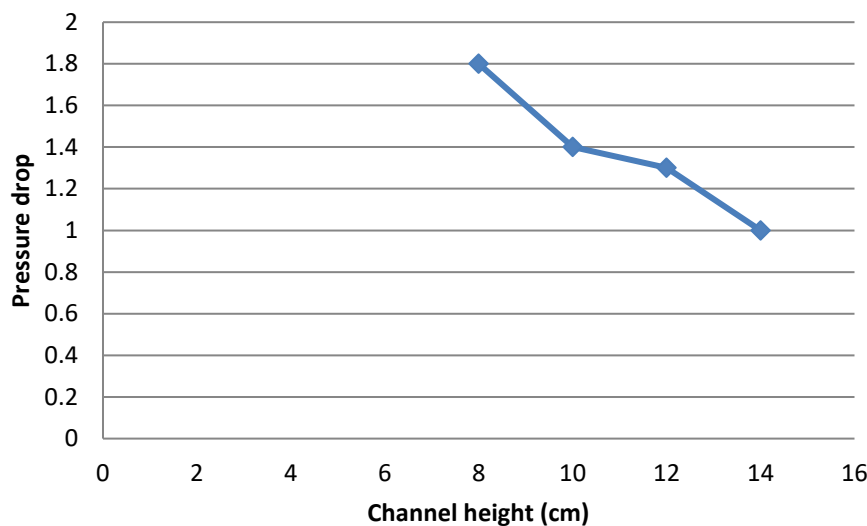


Figure 15. Pressure drop versus channel height

Thermo-hydraulic performance

The thermo-hydraulic performance parameter was used to evaluate the overall performance of the solar air heater by considering both heat transfer enhancement and pressure losses. This parameter provides a comprehensive assessment of system efficiency by combining thermal and hydraulic effects. The calculated values indicate that the thermo-hydraulic performance increases with channel height up to 12 cm. At this configuration, the system achieves the highest performance value, indicating an optimal balance between temperature rise and pressure drop. The improved performance at this height is attributed to enhanced airflow mixing and sufficient contact between the air and heated surfaces while maintaining moderate flow resistance. Although the 14 cm channel height produces the lowest pressure drop, the reduction in heat transfer limits its overall performance. Conversely, smaller channel heights provide stronger heat transfer but result in excessive hydraulic losses. Therefore, based on the thermo-hydraulic performance criterion, the channel height of 12 cm represents the optimal configuration for the present solar air heater design (Figure 16).

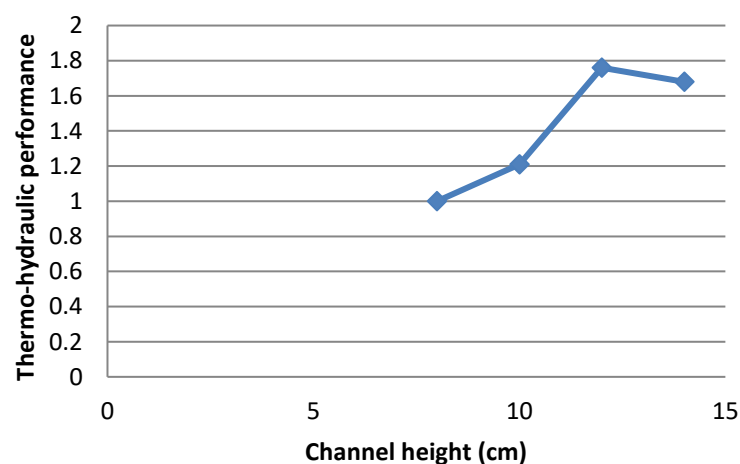


Figure 16. Thermo-hydraulic performance versus channel height

Figure 16 shows the variation of thermo-hydraulic performance at different channel heights as shown in Figure 17. The performance increases with channel height and reaches its maximum value at 12 cm. Beyond this value, a slight decrease is observed at 14 cm, indicating that 12 cm provides the optimal channel height for the SAH configuration.

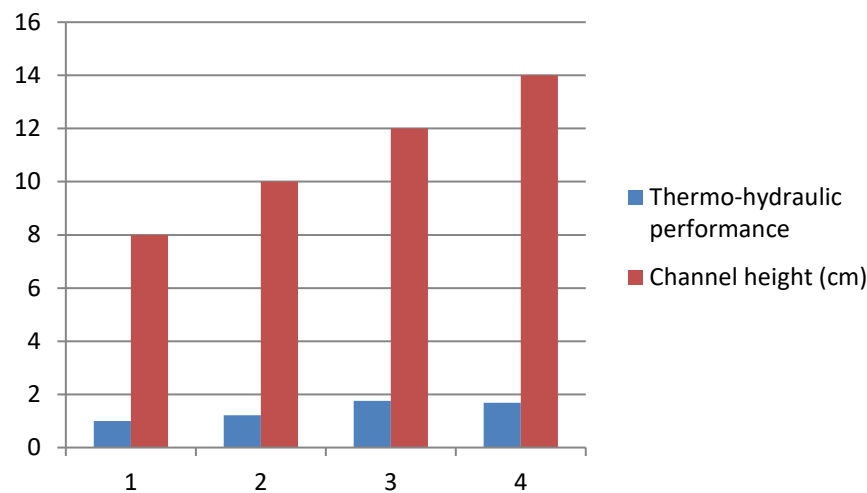


Figure 17. Thermo-hydraulic performance at different channel heights

CONCLUSIONS

A computational fluid dynamics investigation was conducted to analyze the effect of channel height (aspect ratio) on the thermo-hydraulic performance of a packed-bed solar air heater filled with spherical aluminum balls. Four channel heights (8, 10, 12, 14 cm) were examined under identical operating conditions using ANSYS 2023 R1. The results revealed that channel height significantly affects both thermal and hydraulic characteristics. The temperature rise varied between 13.2 and 19.4 °C, while the pressure drop ranged from 1.0 to 1.8 Pa. Increasing the channel height enhanced flow development and increased the Nusselt number from 26.5 to 42.4, whereas the friction factor decreased from 0.013 to 0.008, indicating reduced flow resistance. Among the investigated configurations, the 12 cm channel height provided the most favorable thermo-hydraulic performance. At this configuration, the system achieved the highest thermal efficiency of approximately 85 %, with a moderate pressure drop of 1.3 Pa and a temperature rise of 19.4 °C, indicating an optimal balance between heat transfer enhancement and hydraulic losses. These findings confirm that optimizing the channel height (aspect ratio) plays a critical role in improving the overall efficiency of packed-bed solar air heaters.

Future work may investigate a hybrid packed-bed configuration in which the first and third rows of spherical aluminium elements remain fixed, while the intermediate row is allowed to rotate. Such a configuration is expected to intensify airflow mixing and periodically disrupt the thermal boundary layer, thereby enhancing heat transfer while maintaining relatively low pressure losses. Further optimization of the rotating elements, their rotational speed, and spatial arrangement may provide additional improvements in the thermo-hydraulic performance of packed-bed solar air heaters.

ACKNOWLEDGMENTS

The authors would like to thank the Energy and Renewable Energies Department / Electromechanical Engineering College/ University of Technology for the technical support in conducting this work.

NOMENCLATURE

A	absorber area	$[m^2]$
AR	channel aspect ratio	$[-]$
C_p	specific heat capacity of air	$[J/(kg \cdot K)]$
D_h	hydraulic diameter	$[m]$
f	friction factor	$[-]$
f_0	friction factor of the reference case	$[-]$

h	convective heat transfer coefficient	$[\text{W}/(\text{m}^2 \cdot \text{K})]$
H	channel height	$[\text{m}]$
I	solar heat flux	$[\text{W}/\text{m}^2]$
k	thermal conductivity of air	$[\text{W}/(\text{m} \cdot \text{K})]$
L	channel length	$[\text{m}]$
$\dot{m}$	air mass flow rate	$[\text{kg}/\text{s}]$
Nu	Nusselt number	$[-]$
Nu_0	Nusselt number of the reference case	$[-]$
p_i	inlet pressure	$[\text{Pa}]$
p_{ou}	outlet pressure	$[\text{Pa}]$
Q_u	useful heat gain	$[\text{W}]$
Re	Reynolds number	$[-]$
T_i	inlet air temperature	$[\text{°C}]$
T_{ou}	outlet air temperature	$[\text{°C}]$
u	inlet air velocity	$[\text{m}/\text{s}]$
W	channel width	$[\text{m}]$
Δp	pressure drop	$[\text{Pa}]$
ΔT	temperature difference	$[\text{°C}]$

Greek letters

η	thermal efficiency	$[-]$
μ	dynamic viscosity	$[\text{kg}/(\text{m} \cdot \text{s})]$
ρ	air density	$[\text{kg}/\text{m}^3]$

REFERENCES

1. A. E. A. Abdulmajeed, A. D. Tuncer, A. Coşgun, and A. Güngör, Numerical analysis of thermal performance and pressure loss in parallel-pass solar air heaters with diverse absorber plate designs, *Sadhana*, Vol. 51, No. 1, 2026, <https://doi.org/10.1007/s12046-025-03007-2>.
2. N. Van Hap, P. T. Nhan, and N. M. Phu, Numerical and experimental investigation of the solar air heater with latent heat storage and fin, *Sci. Rep.*, Vol. 15, 2025, <https://doi.org/10.1038/s41598-025-15959-w>.
3. S. Chamarthi and S. Singh, A comprehensive review of experimental investigation procedures and thermal performance enhancement techniques of solar air heaters, *Int. J. energy Res.*, Vol. 45, No. 4, 2020, <https://doi.org/10.1002/er.6255>.
4. B.-H. An, K.-A. Moon, S.-B. Kim, and H.-U. Choi, Analysis of Heat Transfer and Fluid Flow in a Solar Air Heater with Sequentially Placed Rectangular Obstacles on the Fin Surface, *Energies*, Vol. 18, No. 14, 3811, 2025, <https://doi.org/10.3390/en18143811>.
5. S. R. Ghanem and A. C. Bhosale, Exergy-energy efficiency and environmental- economic analysis of honeycomb-shaped artificial roughness in solar air heater, *Discov. energy*, Vol. 5, 2025, <https://doi.org/10.1007/s43937-025-00089-1>.
6. R. Khatri, S. Goswami, M. Anas, S. Sharma, S. Agarwal and S. Aggarwal, Performance evaluation of an arched plate solar air heater with porous aluminum wiremesh cylindrical fins, *Energy Reports*, Vol. 6, pp 627–633, 2020, <https://doi.org/10.1016/j.egy.2020.11.177>.
7. H. S. Arunkumar, K. V. Karanth, and S. Kumar, Review on the design modifications of a solar air heater for improvement in the thermal performance, *Sustain. Energy Technol. Assessments*, Vol. 39, 100685, 2020, <https://doi.org/10.1016/j.seta.2020.100685>.
8. S. M. Salih, J. M. Jalil, and S. E. Najim, Comparative study of novel solar air heater with and without latent energy storage, *J. energy storage*, Vol. 32, 101751, 2020, <https://doi.org/10.1016/j.est.2020.101751>.

9. G. N. Uzakov, S. M. Shomuratova, and B. M. Toshmamatov, Study of a solar air heater with a heat exchanger–accumulator, *IOP Conf. Ser. Earth Environ. Sci.*, Vol. 723, No. 5, 052013, 2021, <https://doi.org/10.1088/1755-1315/723/5/052013>.
10. H. Kazemi Moghadam, S. S. Mousavi Ajarostaghi, and S. Poncet, Numerical modeling of an innovative arc shape rib based solar air heater, *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.*, Vol. 235, No. 24, pp 7992–8012, 2021, <https://doi.org/10.1177/09544062211039541>.
11. W. Siddique, A. Raheem, M. Aqeel, S. Qayyum, T. Salamen, K. Waheed, and K. Qureshi, Evaluation of thermal performance factor for solar air heaters with artificially roughened channels, *Arch. Mech. Eng.*, Vol. 68, No. 2, pp 195–225, 2021, <https://doi.org/10.24425/ame.2021.137048>.
12. S. Nashee and K. S. Mushatet, Performance study on turbulent heat transfer using rectangular air duct integrated with continuous and intermittent ribs turbulators, *Therm. Sci.*, Vol. 29, No. 2A, pp 955–967, 2025, <https://doi.org/10.2298/TSCI240430214N>.
13. K. K. I. Al-Chlahawi, B. H. Alyas, and A. A. Badr, CFD Based Numerical Performance Assessment of a Solar Air Heater Duct Roughened by Transverse - Trapezoidal Sectioned Ribs, *Int. J. heat Technol.*, Vol. 41, No. 5, pp 1273–1281, 2023, <https://doi.org/10.18280/ijht.410517>.
14. M. M. Addini and S. A. Gandjalikhan Nassab, Performance analysis of an efficient new-design solar air heater by CFD method, *J. Serbian Soc. Comput. Mech.*, Vol. 17, No. 1, pp 1–24, 2023, <https://doi.org/10.24874/jsscm.2023.17.01.01>.
15. H. Ali, M. R. Rasani, Z. Harun, and M. A. Shahid, CFD based design optimization of dimples induced on Blended Wing Body airframe using the Taguchi method, *PLoS One*, Vol. 20, No. 4, e0320885, 2025, <https://doi.org/10.1371/journal.pone.0320885>.
16. R. Shaik, E. Punna, and S. K. Gugulothu, Numerical investigation on comparative performance assessment of solar air heater with different artificial roughness elements in a triangular duct, *J. Therm. Eng.*, Vol. 11, No. 1, pp 49–61, 2025, <https://doi.org/10.14744/thermal.0000903>.
17. A. C. Ferreira and Â. Silva, Application of a Costing Methodology to Estimate Capital Costs of Solar Thermal Systems in Residential Portuguese Context, *Int. J. Sustain. Energy Plan. Manag.*, Vol. 26, pp 33–46, 2020, <https://doi.org/10.5278/ijsepm.3483>.
18. W. Bowley and R. Evins, Energy system optimization including carbon-negative technologies for a high-density mixed-use development, *Int. J. Sustain. Energy Plan. Manag.*, Vol. 31, pp 211–225, 2021, <https://doi.org/10.5278/ijsepm.5843>.
19. V. P. Singh, S. Jain, A. Karn, A. Kumar, G. Dwivedi, C. S. Meena, N. Dutt, and A. Ghosh, Recent Developments and Advancements in Solar Air Heaters : A Detailed Review, *Sustain.*, Vol. 14, No. 19, 2022, <https://doi.org/10.3390/su141912149>.
20. K. Mausam and R. Rohatgi, Analysis of advancement in Solar air heater (SAH) heat transfer expansion methods: A crisp review, *Int. Conf. Futur. Sustain. Asp. Eng. Technol. (FSAET 2020)*, Vol. 1116, No. 1, 012050, 2021, <https://doi.org/10.1088/1757-899x/1116/1/012050>.
21. A. Boulemtafes-Boukadoum and A. Benzaoui, CFD based analysis of heat transfer enhancement in solar air heater provided with transverse rectangular ribs, *Energy Procedia*, Vol. 50, pp 761–772, 2014, <https://doi.org/10.1016/j.egypro.2014.06.094>.
22. F. Menasria, M. Zedairia, and A. Moumami, Numerical study of thermohydraulic performance of solar air heater duct equipped with novel continuous rectangular baffles with high aspect ratio, *Energy*, Vol. 133, pp 593–608, 2017, <https://doi.org/10.1016/j.energy.2017.05.002>.

23. A. M. Ahmed, Investigation of solar air heater optimization with spherical Rib, *Edelweiss Appl. Sci. Technol.*, Vol. 9, No. 5, pp 2656–2669, 2025, <https://doi.org/10.55214/25768484.v9i5.7581>.
24. S. Sharma, R. Maithani, and R. K. Das, CFD Based Performance Evaluation of Solar Air Heater by using Centerline Perforated Sine Wave Baffles, *Evergreen*, Vol. 11, No. 2, pp 862–871, 2024, <https://doi.org/10.5109/7183368>.
25. R. H. Pletcher, J. C. Tannehill, and D. A. Anderson, *Computational Fluid Mechanics and Heat Transfer*, 3rd ed. Boca Raton, FL, USA: CRC Press, 2012, <https://doi.org/10.1201/b12884>.
26. T. L. Bergman, A. S. Lavine, F. P. Incropera, and D. P. DeWitt, *Fundamentals of Heat and Mass Transfer*, 8th ed. Hoboken, NJ, USA: Wiley, 2017.
27. J.-C. Han and L. M. Wright, *Experimental Methods in Heat Transfer and Fluid Mechanics*, 1st ed. Boca Raton, FL, USA: CRC Press, 2020, <https://doi.org/10.1201/9781003021179>.
28. H. Ebadi, A. Cammi, and L. Savoldi, Numerical Investigation of the Performance of a Solar Air Heater Equipped with Packed Beds, EuroSun 2022, Kassel, Germany, 2022, <https://doi.org/10.18086/eurosun.2022.12.05>.



Paper submitted: 14.03.2026

Paper revised: 09.08.2026

Paper accepted: 23.08.2026