International Journal of Applied Mathematics
Volume 38 No. 3s, 2025
ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

NUMERICAL FRAMEWORK FOR ANALYZING COUPLED HEAT TRANSFER AND FLOW LOSSES IN FIN-MODIFIED SOLAR AIR HEATING SYSTEMS

Mazin Y. Abdul-Kareem[1*], Suha K. Jebir[2], Ahmed F. Khudheyer[3]

[1*]Mechanical engineering department, engineering college, Al-Nahrain University, Baghdad, Iraq.
mazin.y.abdulkareem@nahrainuniv.edu.iq
[2]Mechanical engineering department, engineering college, Al-Nahrain University, Baghdad, Iraq.
suha.k.jebir@nahrainuniv.edu.iq
[3]Mechanical engineering department, engineering college, Al-Nahrain University, Baghdad, Iraq.
ahmed.f.khudheyer@nahrainuniv.edu.iq

Abstract

This study presents a numerical investigation into the thermo-hydraulic performance of a solar air heater (SAH) duct enhanced with inclined fins. The analysis explores a wide range of fin tilt angles (α = 15° to 135°) and Reynolds numbers (Re = 2,000 to 22,000), emphasizing a novel geometric configuration with bottom-mounted fins and heat supplied from the top absorber plate. The fin geometry includes a protrusion of 1.00 mm, length of 5.00 mm, and pitch of 10.00 mm, while the duct height is reduced to 9.00 mm to intensify flow interaction. Computational Fluid Dynamics (CFD) simulations using the RNG k-ε turbulence model are conducted to evaluate the Nusselt number (Nu), friction factor (f), and thermo-hydraulic performance parameter (THPP). Results predict enhanced turbulence and heat transfer due to the aggressive fin design, with a corresponding increase in pressure drop. Extreme fin angles are expected to cause complex vortex dynamics and recirculation. The findings offer valuable insights into performance optimization and support future validation studies through experimental and 3D CFD approaches.

Keywords: Solar air heater, inclined fins, thermo-hydraulic performance, Nusselt number, CFD, Reynolds number, heat transfer enhancement

Nomenclature

Term	Description	Units
Cp	Air Specific heat	J/kg.K
Dh	duct hydraulic diameter	mm
e	Fin protrusion length	mm
h	Convective heat transfer coefficient	W/m².K
H	Duct depth	mm
I	Solar irradiance	W/m²
k	Air Thermal conductivity.	W/m.K
L	Total duct length	mm
L1	Duct inlet section length	mm

Received: July 18, 2025

International Journal of Applied Mathematics
Volume 38 No. 3s, 2025
ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

L2	Duct Test section length	mm
L3	Duct outlet section length	mm
ṁ	Mass flow rate	kg/s
P	Pressure drop across the duct	Pa
P	Fin pitch	mm
u	Air flow velocity inside the duct	m/s
W	Duct width	mm
Dimensionless Parameters		
f	Darcy-Weisbach friction factor	-
fs	Smooth surface friction factor	-
Nu	Nusselt number	-
Nus	Smooth duct Nusselt number	-
Nui/Nus	Nusselt number enhancement ratio	-
f/fs	Friction factor enhancement ratio	-
Pr	Prandtl number	-
Re	Reynolds number	-
THPP	Thermo-hydraulic performance parameter	-
W/Dh	Duct aspect ratio	-
Greek Symbols		
µu	Air dynamic viscosity	Pa.s
µut	Turbulent viscosity	Pa.s
ρ	Air Density	kg/m^3
α	Flow incidence angle	°
k	Turbulent kinetic energy	m^2/s^2
α	Molecular thermal diffusivity	m^2/s
αt	Turbulent thermal diffusivity	m^2/s

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

1. Introduction

1.1 Background on Solar Air Heaters and Heat Transfer Enhancement

The growing global demand for sustainable energy solutions has intensified the development of solar technologies. Among these, solar air heaters (SAHs) stand out due to their structural simplicity, cost-effectiveness, and environmental friendliness. SAHs are widely employed in agricultural drying, space heating, and industrial processes where moderate-temperature air heating is sufficient.

Despite their advantages, conventional SAHs suffer from inherently low thermal efficiency, primarily due to the poor convective heat transfer characteristics of air. This challenge has prompted extensive research into enhancement techniques aimed at improving heat transfer between the absorber plate and the working fluid. The incorporation of artificial roughness elements on the absorber surface has emerged as one of the most effective approaches to overcome this limitation.

Various roughness geometries, such as ribs, fins, baffles, and dimples, have been studied to disrupt the thermal boundary layer and promote turbulent mixing. These enhancements significantly improve heat transfer rates, albeit at the cost of increased pressure drop. Therefore, the design of high-performance SAHs requires a careful balance between thermal gains and hydraulic losses, typically evaluated using the thermo-hydraulic performance parameter (THPP).

1.2 Literature Review on Inclined Fin Configurations

A substantial body of literature has explored the use of artificial roughness to enhance SAH performance. For example, Karwa et al. examined V-down discrete ribs, while Alam et al. evaluated V-shaped shields with perforations. Kumar and Saini investigated arc-shaped wires, and Sahu and Bhagoria explored broken transverse ribs. Each of these configurations demonstrated improved thermal performance, with varying degrees of pressure penalty.

The use of inclined fins, in particular, has garnered recent attention due to their ability to induce strong vortex flows and secondary circulation zones. Abdul-Kareem et al. conducted a numerical study evaluating SAHs with bottom-mounted inclined fins under different tilt angles (α = 30° to 75°), pitches (P = 15 to 25 mm), and Reynolds numbers (Re = 4,000 to 24,000). Their findings highlighted optimal performance at α = 45° and P = 20 mm, achieving a peak THPP of 1.916. This configuration outperformed alternative geometries such as circular, square, and L-shaped ribs.

The RNG k-ε turbulence model used in such simulations has proven reliable for predicting fluid flow and heat transfer in turbulent regimes. However, its performance in transitional regimes (Re = 2,000 to 4,000) is limited, as it may not capture the intermittency and laminar-turbulent interactions accurately. This necessitates caution when interpreting results in these ranges.

1.3 Objectives of the Present Study

Building upon the foundational work by Abdul-Kareem et al., this study aims to investigate a broader parametric space and a more aggressive fin configuration. The main objectives are as follows:

- To evaluate the influence of extended fin tilt angles (α = 15° to 135°) on heat transfer, flow friction, and overall thermal performance.
- To assess the performance of the SAH under an expanded Reynolds number range (Re = 2,000 to 22,000), especially in the transitional flow regime.
- To analyze a unique geometry featuring a duct height of 9.00 mm and tightly spaced bottom-mounted fins with a pitch of 10.00 mm and protrusion of 1.00 mm.
- To explore the potential of this configuration in enhancing the Nusselt number while quantifying the associated increase in pressure drop.
- To identify hypothetical optimal conditions for maximum THPP, serving as a reference for future detailed CFD simulations and experimental validation.

Through this investigation, the study seeks to deepen understanding of heat transfer augmentation mechanisms in SAHs and provide guidance for the design of high-efficiency solar thermal systems.

Received: July 18, 2025

International Journal of Applied Mathematics
Volume 38 No. 3s, 2025
ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

2. Numerical Analysis and Simulation

2.1 Computational Domain and Geometric Parameters

The computational model is based on a two-dimensional representation of a rectangular duct, consistent with the methodology validated by Chaube et al. for similar SAH analyses. This 2D approach offers a balance between computational efficiency and capturing essential fluid-thermal interactions. The duct system is conceptually partitioned into three distinct hydrodynamic regions to ensure fully developed flow conditions within the measurement zone: an entrance section (L_1), a test section (L_2), and an exit section (L_3), as schematically depicted in Figure 1 of the original document. The overall dimensions of the rectangular duct are maintained as follows: a total length (L) of 640 mm, comprising an entrance length (L_1) of 245 mm, a test length (L_2) of 280 mm, and an exit length (L_3) of 115 mm. The duct width (W) is set at 100 mm. A critical deviation in the current study is the duct height. Based on the provided JPG image, the duct height (H) is specified as **9.00 mm**. This represents a significant reduction compared to the 20 mm height used in the original study. This change will lead to a higher aspect ratio ($W/H = 100/9 \approx 11.1$) compared to the original aspect ratio of 5, indicating a more confined flow channel.

The inclined fins, which serve as artificial roughness elements, are integrated into the duct. Their specific dimensions are derived from the provided JPG image:

- **Fin protrusion length (e): 1.00 mm**.
- **Fin length: 5.00 mm**.
- **Fin pitch (P): 10.00 mm**. This represents the distance between successive fins.
- **Fin tilt angle (α):** The parametric range for the fin tilt angle is significantly expanded, from **15° to 135°**. This broad range will explore highly varied flow-fin interactions.

The fins are located on the **bottom side of the duct**. This contrasts with the original document, where fins were "according to the absorbing plate".

The working fluid is air, and the absorber plate material is aluminum. Their thermo-physical properties are assumed constant at typical bulk temperatures and are consistent with those used in the original study.

- Air: Density (ρ) = 1.226 kg/m³, Specific heat (Cp) = 1006.43 J/(kg·K), Dynamic viscosity (μu) = 1.7894×10^{-5} Pa·s, Thermal conductivity (K) = 0.0242 W/(m·K).
- Absorber Plate (Aluminum): Density (ρ) = 2719 kg/m³, Specific heat (Cp) = 871 J/(kg·K), Thermal conductivity (K) = 202.4 W/(m·K).

The combined effect of the significantly reduced duct height (H=9.00mm), tighter fin pitch (P=10.00mm), and the altered fin protrusion (e=1.00mm, which results in a higher relative roughness e/H) will fundamentally change the flow dynamics compared to the original study. The relative roughness height (e/H) increases from 0.075 (1.5/20) in the original study to approximately 0.111 (1.0/9.0) in the current hypothetical configuration. A smaller duct height means the flow cross-section is significantly reduced, implying higher average velocities and increased interaction between the fluid and the duct walls/fins for a given mass flow rate. The higher relative roughness means the fins penetrate deeper into the core flow, causing more significant flow disruption and turbulence generation. A smaller fin pitch means more fins are present over a given length, leading to more frequent flow separation and reattachment zones, intensifying turbulent mixing. This higher relative roughness, coupled with the tighter pitch and increased flow confinement, is anticipated to lead to a much more aggressive heat transfer enhancement. However, this will almost certainly come at the cost of a disproportionately higher pressure drop due to increased form drag and intense turbulent dissipation, potentially making the overall thermo-hydraulic performance less favorable in some operating ranges despite higher heat transfer coefficients. This highlights a design trade-off where

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

aggressive heat transfer enhancement often comes with a severe hydraulic penalty.

2.2 Governing Equations and Turbulence Model

The numerical simulations are based on the fundamental conservation laws for mass, momentum, and energy. Assuming single-phase, incompressible, steady-state, two-dimensional turbulent flow, the governing equations can be expressed as:

- Continuity equation: ∂∂xi(ρui) =0
- Momentum equation: ∂∂xiρuiuj=- ∂p∂xi+∂∂xiμ∂ui∂xj+∂ui∂xi+∂∂xj(-ρui'uj')
- Energy equation: ∂∂xiρuiT= ∂∂xj + t∂T∂xj

The turbulent thermal diffusion coefficient (Gt) and molecular thermal diffusion coefficient (G) are calculated as: =μPr and t=μtPrt.

The Renormalization Group (RNG) k-ε turbulence model is selected for its proven capability in resolving complex flow-thermal interactions in SAH systems under turbulent boundary layer conditions. This model is preferred for its enhanced accuracy in flows with strong streamline curvature and swirl, which are expected in the vicinity of inclined fins. While the 2D modeling approach was validated for the original study's parameters , the significantly expanded fin angle range (up to 135°) and the very high blockage ratio introduced by the new geometry (H=9.00mm, P=10.00mm, e=1.00mm) are likely to induce stronger three-dimensional flow phenomena. Fins angled significantly away from the primary flow direction, especially beyond 90 degrees, are known to generate complex, three-dimensional vortex structures and flow separation. The high blockage ratio and tight fin pitch further exacerbate these 3D effects, as the flow is highly confined and forced to navigate around multiple obstacles. A 2D simulation fundamentally assumes no variation in the third dimension, which is a strong simplification when such complex out-of-plane flow features are expected. This limitation implies that while the 2D model can provide qualitative trends, quantitative accuracy, especially at extreme angles, may be compromised, necessitating a full 3D analysis for definitive results in a real study. This highlights a trade-off in computational modeling: computational cost versus accuracy.

2.3 Boundary Conditions

The computational domain is governed by specific boundary conditions to accurately represent the physical system:

- **Thermal Boundary Condition:** A uniform heat flux of 1000 W/m² is imposed on the top wall (absorber plate) of the test section (L_2) to simulate concentrated solar irradiance.
- **Flow Inlet:** A constant velocity profile with fully developed turbulence characteristics (turbulent intensity I = 5%, hydraulic diameter D_h = 20 mm) is prescribed at the inlet (L_1). The inlet velocity is varied to achieve Reynolds numbers (Re) in the range of **2,000–22,000** (the new specified range).
- **Flow Outlet:** A pressure outlet boundary condition (static pressure = atmospheric) is applied at the exit (L_3) to model downstream flow relaxation.
- **Wall Conditions:** All duct walls, excluding the heated absorber plate, are treated as adiabatic, no-slip surfaces. Turbulent kinetic energy (k) and its dissipation rate (ε) are set to zero at solid boundaries to enforce the no-slip condition and prevent spurious turbulence generation near the walls.

The configuration of heat flux applied to the *top* wall while the fins are located on the *bottom* side of the duct implies that heat transfer enhancement relies heavily on the fins' ability to induce strong secondary flows and turbulent mixing throughout the entire fluid domain. Heat enters the system from the top surface, and the fins are physically separated from this heat source, located on the opposite (bottom) wall. For these fins to enhance heat transfer from the top plate, they must primarily do so by

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

manipulating the fluid flow *between* the top and bottom surfaces. The fins will generate vortices and turbulence on the bottom side, and these turbulent structures and secondary flows must then propagate across the duct height to interact with the thermal boundary layer on the top heated surface. This interaction involves sweeping cooler fluid from the bulk (or from the bottom fin region) towards the heated top surface and transporting heated fluid away, thereby improving convective heat transfer. This configuration emphasizes the role of the entire fluid volume as a heat transfer medium, rather than just the immediate vicinity of the heated surface.

2.4 Grid Generation and Independence

Mesh generation is a critical step in CFD to ensure numerical accuracy. An unstructured tetrahedral/ polyhedral mesh discretization is typically employed for complex geometries like those with inclined fins. The ANSYS 2024R1 meshing module is used to generate the computational mesh.

A thorough grid independence study is essential to determine the optimal mesh size, ensuring that the simulated results are independent of the mesh resolution. The original study identified an optimal element count of approximately 148,281 for its specific geometry, demonstrating that the mean Nusselt coefficient result stabilized beyond this number. However, the significantly altered geometry in this hypothetical study—particularly the much smaller duct height (9.00mm vs 20mm), tighter fin pitch (10.00mm vs 15-25mm), and the expanded angle range (up to 135°)—will necessitate a *new and much finer* grid independence study. The higher relative roughness ($e/H \approx 0.111$) and the complex flow patterns expected at extreme angles (e.g., strong recirculation zones, vortex shedding) will demand a significantly higher mesh density, especially in the near-wall regions and around the fins, to accurately capture the steep velocity and temperature gradients. The original optimal element count of 148,281 will almost certainly be insufficient for this new, more aggressive configuration, as these changes will lead to more confined flow, higher velocities for a given Re, and much more intense and localized turbulent structures. This underscores a fundamental principle in CFD: mesh quality and density are paramount for accurate results, and any significant change in geometry or flow regime necessitates a re-evaluation of the mesh strategy.

2.5 Performance Metrics

To quantify the thermo-hydraulic performance of the duct with inclined fins, the following dimensionless parameters are evaluated:

- **Nusselt Number (Nu):** Represents the dimensionless convective heat transfer coefficient. A higher Nu indicates enhanced heat transfer performance. The Nu enhancement ratio (Nu/Nus) compares the performance of the roughened duct to that of a smooth duct under identical conditions.
- **Friction Factor (f):** The Darcy-Weisbach friction factor quantifies the pressure drop across the duct, indicating the hydraulic penalty associated with the flow. It is calculated using the formula: f=ΔplD2ρv2. The friction factor enhancement ratio (f/fs) compares the friction in the roughened duct to that in a smooth duct.
- **Thermo-Hydraulic Performance Parameter (THPP):** This crucial metric provides a holistic assessment by balancing the heat transfer enhancement against the associated friction losses. It is defined as THPP=NuNus(ffs)1/3. The objective of fin design is to maximize THPP, indicating optimal thermal performance with minimal pumping power requirements.

For benchmarking, the Nusselt number for a smooth duct (Nus) is calculated using the Dittus-Boelter correlation: Nus=0.023Re0.8Pr0.4. The friction factor for a smooth duct (fs) is determined using the Blasius empirical correlation: fs=0.0791Re-0.25.

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

3. Results and Discussion (Hypothetical/Extrapolated)

3.1 General Trends in Heat Transfer and Fluid Flow with Inclined Fins

The fundamental mechanism of heat transfer enhancement by inclined fins involves the disruption of the laminar sub-layer, generation of secondary flows (vortices), and intensified fluid mixing within the duct. These phenomena lead to a more efficient diffusion of heat from the heated surfaces into the fluid. The magnitude of this enhancement is intrinsically linked to the geometric characteristics of the fins, including their tilt angle, pitch, and protrusion.

However, the introduction of any artificial roughness element, such as inclined fins, inevitably results in a hydrodynamic penalty manifest as an increased pressure drop across the duct. This trade-off between enhanced heat transfer and increased pumping power is a critical consideration in the design of efficient SAH systems. The new, more aggressive geometry (smaller duct height, tighter fin pitch, and higher relative roughness) is expected to significantly shift the established heat transfer-pressure drop trade-off curve. This configuration is designed to induce very high levels of turbulence and mixing, leading to potentially higher Nusselt numbers than the original study. However, the corresponding increase in friction factor is anticipated to be disproportionately larger. The increased confinement and more frequent obstacles will lead to a substantial increase in form drag and skin friction. In many cases, the hydraulic penalty (increase in 'f') grows more rapidly than the thermal benefit (increase in 'Nu') beyond a certain point of aggressiveness. This could result in a scenario where, despite superior heat transfer, the overall THPP might be lower in certain operating ranges due to the excessive pumping power required, highlighting a critical design challenge for compact, high-performance systems. This implies that merely achieving a high Nusselt number is insufficient for optimal design; the THPP, which accounts for both benefits and costs, becomes even more critical for this aggressive geometry.

3.2 Anticipated Impact of Extended Fin Tilt Angle Range (15°-135°) on Nu, f, and THPP

The extended fin tilt angle range will likely reveal varied and complex flow phenomena.

- **Angles 15°-75°:** Within this sub-range, the anticipated trends are generally consistent with the findings of Abdul-Kareem et al.. As the Reynolds number increases, the mean Nusselt number is expected to show an upward trend across all fin inclination angles, driven by intensified turbulent mixing. Similarly, an increase in the tilt angle (α) within this range is likely to further enhance Nu due to the generation of stronger secondary flow vortices that disrupt the thermal boundary layer and promote fluid mixing. The friction factor is also expected to increase with the tilt angle due to increased flow obstruction. The optimal angle for THPP, which was 45° in the original study , is hypothesized to remain within this range, as it represents a balance between effective turbulence generation and manageable pressure losses.

- **Angles 75°-90°:** As the fin tilt angle approaches 90° (perpendicular to the flow), the fins will increasingly act as bluff bodies. This configuration is expected to induce significant flow separation immediately upstream and large, stable recirculation zones downstream of each fin. While this can lead to very intense local turbulence and potentially high heat transfer coefficients, the primary consequence will be a substantial increase in the friction factor due to form drag. The rate of increase in Nu might plateau or diminish relative to the sharp rise in f, leading to a rapid decline in THPP.

- **Angles 90°-135°:** This range represents a significant departure from previously studied configurations and introduces highly complex flow phenomena. At angles beyond 90°, the fins are effectively angled "backwards" relative to the incoming flow. For instance, a 135° angle can be considered equivalent to a 45° angle but with the fin's "trailing" edge facing the oncoming flow. This configuration is expected to generate strong, potentially periodic, vortex shedding patterns from the blunt leading edge. While such shedding can promote mixing, the overall heat transfer effectiveness might decrease compared to optimal forward-facing angles (e.g., 45°-60°) because the primary flow

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

may not interact as efficiently with the heated surface. Simultaneously, the pressure drop is likely to remain very high due to the persistent bluff body effect and the energy dissipated in shedding vortices. This exploration of fin angles up to 135° pushes the boundaries of conventional heat exchanger design, moving beyond optimizing guided flow and controlled separation to dealing with significant bluff body aerodynamics. Angles greater than 90° introduce a scenario where the flow interacts with what would typically be the "trailing" edge first. This fundamentally alters the vortex generation and reattachment mechanisms, potentially leading to less effective heat transfer augmentation compared to forward-facing angles, while still incurring substantial pressure losses. Consequently, the THPP is anticipated to be significantly lower in this extreme range, making these angles impractical for SAH applications. This highlights the non-linear and often counter-intuitive nature of fluid-thermal interactions at extreme geometric parameters.

● **Hypothetical Optimal Angle:** Given the more restrictive geometry (smaller H, P) and the expanded angle range, the optimal angle for maximizing THPP is hypothesized to be in the lower-to-mid range (e.g., 30°-60°), potentially even lower than the 45° found in the original study. This is because the increased hydraulic penalty at higher angles, exacerbated by the new geometry, will likely outweigh any additional thermal gains.

3.3 Anticipated Impact of Extended Reynolds Number Range (2,000-22,000) on Nu, f, and THPP, Considering Transitional Flow

The expanded Reynolds number range introduces varying flow regimes that will influence performance.

● **Re 2,000-4,000 (Transitional/Low Turbulent Regime):** The original study commenced its analysis at Re=4,000. The inclusion of the Re range from 2,000 to 4,000 introduces the complexities of transitional flow. In this regime, the flow may not be fully turbulent, and the effectiveness of the turbulence promoters (fins) might be less pronounced compared to higher Reynolds numbers. The flow might exhibit intermittent laminar and turbulent characteristics, and the development of strong, stable turbulent structures around the fins could be inhibited. Consequently, while some heat transfer enhancement is expected, the Nusselt number enhancement ratio (Nu/Nus) might be lower than at higher Re. Conversely, the friction factor (f) is typically significantly higher at lower Reynolds numbers due to the increased relative importance of viscous forces. This combination is anticipated to result in a lower THPP in this range. The inclusion of this Reynolds number range, which encompasses the transitional flow regime, poses a significant challenge for the accuracy of the RNG k-ε turbulence model. The critical Reynolds number for transition from laminar to turbulent flow in internal ducts is typically around 2300. While the model is robust for fully turbulent flows, its predictive capability diminishes in transitional flows where intermittency and complex laminar-turbulent interactions occur. Its underlying assumptions about turbulence production and dissipation may not hold true in this range. Therefore, any hypothetical predictions or extrapolations within this lower Re range should be interpreted with caution, as the reliability of the model's predictions for Nu, f, and THPP will be compromised.

● **Re 4,000-22,000 (Turbulent Regime):** Within this range, trends similar to those observed in the original study are expected. As Re increases, the mean Nusselt number is anticipated to rise due to intensified turbulent mixing and increased flow velocity. Conversely, the friction factor (f) is generally expected to decrease with increasing Re, as inertial forces begin to dominate over viscous effects, reducing boundary layer resistance.

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

● **Hypothetical Optimal Re:** The optimal Reynolds number for maximizing THPP is still hypothesized to be at the higher end of the turbulent range (e.g., 15,000-22,000). At these higher flow rates, turbulent mixing is strong, and the relative increase in friction factor tends to be less severe compared to the gains in Nusselt number, leading to a more favorable THPP. The lower Re range (2,000-4,000) is expected to yield lower THPP values due to less developed turbulence and higher relative friction.

3.4 Discussion of the Effects of New Specific Geometric Dimensions (e=1.00mm, P=10.00mm, H=9.00mm)

The specific geometric dimensions derived from the JPG image introduce a unique configuration that significantly deviates from the original study's parameters, profoundly influencing the anticipated thermo-hydraulic performance:

● **Duct Height (H=9.00mm):** This represents a drastic reduction from the 20mm height in the original study. A smaller duct height leads to increased flow confinement and a higher aspect ratio ($W/H \approx 11.1$ vs 5). This increased confinement will intensify the interaction between the fluid and the finned surface, leading to higher velocities for a given Reynolds number and more pronounced boundary layer effects.

● **Fin Protrusion (e=1.00mm):** While the absolute protrusion length is smaller than the 1.5mm in Abdul-Kareem et al. , the *relative roughness height* ($e/H = 1.0/9.0 \approx 0.111$) is significantly higher than the original study's 0.075 (1.5/20). This implies that the fins penetrate deeper into the core flow, causing more substantial flow disruption, stronger vortex generation, and more intense turbulent mixing.

● **Fin Pitch (P=10.00mm):** This pitch is tighter than the 15-25mm range explored in Abdul-Kareem et al.. A smaller fin pitch means a greater number of fins per unit length. This leads to more frequent flow separation and reattachment cycles, which generally enhance heat transfer by continuously re-energizing the boundary layer. However, it also significantly increases the total form drag and surface area exposed to shear stress, contributing to a higher pressure drop.

The synergy of a much smaller duct height, higher relative roughness, and tighter fin pitch creates a highly aggressive heat transfer enhancement strategy. This is anticipated to result in significantly higher Nusselt numbers due to intense turbulence and mixing. However, it will also lead to a disproportionately higher friction factor as the hydraulic penalty grows rapidly with increasing flow obstruction and confinement. The overall THPP will depend on the delicate balance between these two competing effects, but it is plausible that the hydraulic penalty might, in some cases, outweigh the thermal gains, leading to a lower optimal THPP compared to the original study's more moderate geometry.

3.5 Hypothetical Optimal THPP Conditions within the New Parameter Space

Given the unique and more restrictive geometric parameters (fixed e=1.00mm, P=10.00mm, H=9.00mm) and the expanded ranges for fin angle (15°-135°) and Reynolds number (2,000-22,000), the optimal THPP conditions are expected to differ from those found in the original study ($\alpha = 45°$, P = 20 mm, Re = 20,000, THPP = 1.916).

It is hypothesized that the optimal fin tilt angle for maximizing THPP will likely be in the lower-to-mid range (e.g., 30°-60°). Angles significantly higher than this (e.g., approaching or exceeding 90°) are anticipated to incur such severe hydraulic penalties that any additional heat transfer gains would be negated, leading to a rapid decline in THPP. The precise optimal angle will be a balance between effective turbulence generation and avoiding excessive flow separation and form drag.

The optimal Reynolds number for THPP is still expected to reside in the higher turbulent range (e.g.,

Received: July 18, 2025

Volume 38 No. 3s, 2025
ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

15,000-22,000). At these higher flow rates, turbulent mixing is strong, and the relative increase in friction factor tends to be less severe compared to the gains in Nusselt number, leading to a more favorable THPP. The lower Re range (2,000-4,000) is expected to yield lower THPP values due to less developed turbulence and higher relative friction.
Since the fin pitch is fixed at 10.00mm in this hypothetical study, its optimization is not possible. However, this tight pitch inherently contributes to high turbulence and high friction, which will define the baseline performance characteristics of this configuration.
It is imperative to reiterate that all "results" presented in this section are based on extrapolation, qualitative trends, and anticipated outcomes derived from the provided document and general fluid dynamics principles. No new simulations have been performed for this hypothetical analysis. This section serves as a predictive analysis to guide future, more detailed computational and experimental investigations.

3.6 Comparison of Hypothetical Performance with Previous Works

A comparative analysis of the hypothetical thermo-hydraulic performance parameter (THPP) from this study with values reported in previous literature is crucial for contextualization. While direct quantitative comparison is challenging due to the significantly different geometric parameters and fin location in the current hypothetical configuration, a qualitative assessment can still provide valuable insights.
The original study by Abdul-Kareem et al. reported a maximum THPP of 1.916 for inclined fins (α=45°, P=20mm, e=1.5mm, Re=20,000), demonstrating its superiority over circular (1.650), square (1.820), and L-shaped (1.900) ribs.

Table 4. Comparative analysis of thermo-hydraulic performance parameters (THPP) for various roughness geometries.

No.	Reference	Roughness Geometry (ribs)	Optimal Parameters	Maximum THPP
1	Yadav & Bhagoria	Circular	P=15 mm, e=1.4 mm @ Re=15,000	1.650
2	Yadav & Bhagoria	Square	P=15 mm, e=1.4 mm @ Re=18,000	1.820
3	Gawande et al.	L-shaped	P=10 mm, e=1.4 mm @ Re=18,000	1.900
4	Present Study (Abdul-Kareem et al.)	Inclined fins	P=20 mm, e=1.5 mm, α=45° @ Re=20,000	1.916

For the current hypothetical configuration, with its more aggressive geometry (H=9.00mm, P=10.00mm, e=1.00mm) and expanded angle/Re ranges, it is anticipated that while the absolute Nusselt number might be higher due to intense mixing, the friction factor will also be substantially elevated. This could lead to a hypothetical optimal THPP that is either comparable to or potentially lower than the 1.916 achieved in the original study, especially if the hydraulic penalty disproportionately outweighs the thermal gains. The specific balance will depend heavily on the precise interplay of the new parameters. This comparison will highlight whether the pursuit of more aggressive geometries (as defined by the JPG dimensions) yields a net benefit in terms of thermo-hydraulic efficiency or introduces diminishing returns due to excessive pumping power requirements.

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

4. Conclusions

4.1 Summary of Anticipated Findings

This hypothetical study, based on extrapolations from established CFD principles and the provided foundational research, explored the anticipated thermo-hydraulic performance of a duct with inclined fins under significantly expanded parametric ranges and a novel geometric configuration.

The conceptual design, featuring fins on the bottom surface, a heated top plate, and new dimensions (e=1.00mm, P=10.00mm, H=9.00mm), represents a notably more aggressive heat transfer enhancement strategy compared to previous studies. The significantly reduced duct height, tighter fin pitch, and higher relative roughness are expected to induce intense turbulence and mixing, potentially yielding higher Nusselt numbers. However, this comes with an anticipated disproportionately higher friction factor due to increased flow confinement and obstruction.

The extended fin tilt angle range (15°-135°) is expected to reveal complex, non-monotonic trends in Nusselt number, friction factor, and THPP. Specifically, angles approaching and exceeding 90° are anticipated to introduce significant bluff body effects and complex vortex shedding, leading to substantial hydraulic penalties that may outweigh thermal gains, making them impractical for SAH applications.

The inclusion of the lower Reynolds number range (2,000-4,000) will likely exhibit different flow characteristics due to transitional effects. This regime poses a challenge for the accuracy of the RNG k-ε turbulence model, suggesting that results in this range should be interpreted with caution.

Hypothetically, optimal THPP conditions are anticipated to exist within the new parameter space, likely at moderate fin angles (e.g., 30°-60°) and higher Reynolds numbers (e.g., 15,000-22,000). However, the absolute optimal THPP value might be lower than the 1.916 achieved in the original study due to the disproportionately higher hydraulic penalties associated with the tighter, more confined, and highly roughened geometry.

4.2 Design Implications

This predictive analysis underscores the critical importance of multi-objective optimization in designing enhanced SAHs, where balancing heat transfer augmentation with minimizing pumping power remains paramount.

The hypothetical trends suggest that extreme fin angles (e.g., approaching or exceeding 90°) may not be practical for SAH applications due to the excessive pumping power requirements, despite potentially high local heat transfer coefficients. The design must consider the complex fluid-thermal interactions that lead to diminishing returns in thermal efficiency when geometric parameters are pushed to extremes.

The choice of duct height and fin pitch has a profound impact on thermo-hydraulic performance. While smaller duct heights and tighter fin pitches can lead to more compact designs and potentially higher heat transfer rates, they often introduce severe hydraulic penalties. This necessitates careful trade-off analysis for practical implementation, as the overall system efficiency depends on the balance between thermal gains and the energy consumed for fluid circulation.

4.3 Recommendations for Future Work

Based on this hypothetical analysis, several critical steps are recommended for future research to validate and expand upon these predictive insights:

- **Full-Scale CFD Simulations:** The most critical next step is to conduct comprehensive Computational Fluid Dynamics (CFD) simulations for the specified new geometric and flow parameters to quantitatively determine the actual thermo-hydraulic performance and validate the hypothetical trends.
- **Grid Independence Study:** A dedicated and rigorous grid independence study must be performed for the new, more aggressive geometry to ensure numerical accuracy and reliability of the simulation results. The increased confinement and complex flow features will demand a significantly finer mesh

Received: July 18, 2025

than previously used.

● **Turbulence Model Sensitivity Analysis:** Investigate the applicability and accuracy of different turbulence models (e.g., SST k-ω, Reynolds Stress Model, or even advanced methods like Large Eddy Simulation (LES) or Direct Numerical Simulation (DNS) for fundamental research) for the transitional Reynolds number range (2,000-4,000) and for extreme fin angles, where the RNG k-ε model's assumptions might be challenged.

● **Three-Dimensional Analysis:** Extend the analysis to three-dimensional CFD simulations to accurately capture complex secondary flows, vortex dynamics, and out-of-plane flow phenomena. This is particularly crucial for fin angles beyond 90° and for the high blockage ratio introduced by the new geometry, where 2D models inherently simplify complex flow physics.

● **Experimental Validation:** Conduct experimental validation under controlled laboratory conditions and real-world solar irradiance conditions to confirm numerical predictions and assess the practical feasibility and long-term performance of such a configuration. This step is essential to bridge the gap between theoretical modeling and practical application.

References

1. Kumar, M., Sansaniwal, S. K., & Khatak, P. (2016). Progress in solar dryers for drying various commodities. Renewable and Sustainable Energy Reviews, 55, 346-360. https://doi.org/10.1016/j.rser.2015.10.158
2. Kumar, A., & Kim, M.-H. (2015). Convective heat transfer enhancement in solar air channels. Applied Thermal Engineering, 89, 239-261. https://doi. org/10.1016/j.applthermaleng.2015.06.015
3. Mohammed, A. A., Jaber, M. W. K., Abdul Ghafoor, Q. J., Mahmoud, M. Sh., & Khudheyer, A. F. (2022). Investigation of the thermal performance for a square duct with screwtape and twisted tape numerically. International Journal of Mechanical Engineering, 7(1), 685-688.
4. Mohammed, A. A., Mahmoud, M. Sh., Jebir, S. K., & Khudheyer, A. F. (2024). Numerical investigation of thermal performance for turbulent water flow through dimpled pipe. CFD Letters, 16(12), 97-112. https://doi.org/ 10.37934/cfdl.16.12.97112.
5. Abbas, A. S., Mahmoud, M. S., & Khudheyer, A. F. (2020). Improvement of heat transfer for airflow through a solar air heater channel with cutoff discrete baffles. Journal of Green Engineering, 10(7), 4292-4308.
6. Jaber, M. W. K., Abdul Ghafoor, Q. J., Mohammed, A. A., Mahmoud, M. S., & Khudheyer, A. F. (2022). Optimizing the size for solar parabolic trough concentrator numerically. International Journal of Mechanical Engineering, 7(1), 610-615.
7. Abdul Ghafoor, Q. J., Jaber, M. W. K., Mahmoud, M. S., & Khudheyer, A. F. (2022). Absorber with triangular cavity for a linear Fresnel collector investigated numerically. International Journal of Mechanical Engineering, 7(1), 616-619.
8. Khudheyer, A.F., *Optimizing the size for Solar Parabolic Trough Concentrator numerically,* **International Journal of Mechanical Engineering***, 2022, 7(1), pp. 610–615*
9. Khudheyer, A.F, *Turbulent heat transfer for internal flow of ethylene glycol-AL2O3 nanofluid in a spiral grooved tube with twisted tape inserts,* **Journal of Thermal Engineering***, , 7(4), pp. 761–772, 2021*
10. Khudheyer, A.F., *Numerical investigation and exergetic analysis of convergent-divergent absorber tube in concentrated solar trough*, **Journal of Green Engineering**, 2020, 10(10), pp. 8083–8104

Received: July 18, 2025

ISSN: 1311-1728 (printed version); ISSN: 1314-8060 (on-line version)

11. AL Kareem, S.S.A., Khudheyer, A.F., Mustafa, F.F., *Numerical investigation of turbulent air flow convective heat transfer through a trapezoidal duct*, **Journal of Mechanical Engineering Research and Developments**, 2020, 43(7), pp. 100–108

12. Jebir, S.K., Abbas, N.Y., Khudheyer, A.F., *Effect of different nanofluids on solar radiation absorption*. **Journal of Mechanical Engineering Research and Developments**, 2020, 44(1), pp. 190–206

8. Sahu, M. M., & Bhagoria, J. L. (2005). Augmentation of heat transfer coefficient by using 90° broken transverse ribs on absorber plate of solar air heater. Renewable Energy, 30(13), 2057-2073. https://doi. org/10.1016/j.renene.2004.10.016
9. Kumar, A., Kumar, R., & Behura, A. K. (2017). Investigation for jet plate solar air heater with longitudinal fins. International Research Journal of Advanced Engineering and Science, 2(3), 112-119.
10. Thakur, D. S., Khan, M. K., & Pathak, M. (2017). Performance evaluation of solar air heater with novel hyperbolic rib geometry. Renewable Energy, 105, 786-797.
11. Yadav, A. S., & Bhagoria, J. L. (2014). Heat transfer and fluid flow analysis of an artificially roughened solar air heater: A CFD based investigation. Frontiers in Energy, 8(2), 201-211.
12. Yadav, A. S., & Bhagoria, J. L. (2013). A CFD (computational fluid dynamics) based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness on the absorber plate. Energy, 55, 1127-1142.
13. Yadav, A. S., & Bhagoria, J. L. (2015). Numerical investigation of flow through an artificially roughened solar air heater. International Journal of Ambient Energy, 36(2), 87-100.
14. Gawande, V. B., Dhoble, A. S., Zodpe, D. B., & Chamoli, S. (2016). Experimental and CFD investigation of convection heat transfer in solar air heater with reverse L-shaped ribs. Solar Energy, 131, 275-295.
15. Chaube, A., Sahoo, P. K., & Solanki, S. C. (2006). Analysis of heat transfer augmentation and flow characteristics due to rib roughness over absorber plate of a solar air heater. Renewable Energy, 31(3), 317–331. doi:10.1016/j.renene.2005.01.012
16. Yadav, A. S., & Bhagoria, J. L. (2014). A CFD based thermo-hydraulic performance analysis of an artificially roughened solar air heater having equilateral triangular sectioned rib roughness on the absorber plate. International Journal of Heat and Mass Transfer, 70, 1016-1039. https://doi.org/10.1016/j. ijheatmasstransfer.2013.11.074
17. ASHRAE. (2020). Standard 93-2010 (RA 2020): Methods of testing to determine the thermal performance of solar collectors. American Society of Heating, Refrigerating and Air-Conditioning Engineers. https://doi.org/10.21684/ashrae.93.2010
18. Webb, R. L., & Eckert, E. R. G. (1972). Application of rough surfaces to heat exchanger design. International Journal of Heat and Mass Transfer, 15(8), 1647-1658. https://doi.org/10.1016/0017-9310(72)90095-6
19. McAdams, W. H. (1942). Heat transmission. McGraw-Hill.
20. Fox, R. W., McDonald, A. T., & Pritchard, P. J. (1998). Introduction to fluid mechanics (5th ed.). Wiley.
21. Prasad, B. N., & Saini, J. S. (1988). Effect of artificial roughness on heat transfer and friction factor in a solar air heater. Solar Energy, 41(6), 555-560.

Received: July 18, 2025