Al-Rafidain Journal of Engineering Sciences

Journal homepage https://rjes.iq/index.php/rjes

ISSN 3005-3153 (Online)

Mathematical Modeling of Heat Transfer in Energy Conversion Systems

Mohammed Saad Kamil

Research and Technology Azarbaijan Shahid Madani University, Department of Mechanics in Partial Fulfillment, Iran

ARTICLE INFO

Article history:
Received 18 September 2025
Revised 18 September 2025
Accepted 20 October 2025,
Available online 20 October 2025

Keywords:
Parabolic trough collector;
Solar thermal energy;
Heat transfer modeling;
Energy conversion systems;
Thermal efficiency;
Conjugate heat transfer;
Radiation losses;
Numerical simulation;
Python implementation;
Renewable energy.

ABSTRACT

This study presents the development and numerical implementation of a mathematical model for analyzing heat transfer in parabolic trough collectors (PTCs), a widely adopted technology in solar thermal energy conversion. The model integrates conduction, convection, and radiation mechanisms, formulating an energy balance along the absorber tube through which a heat transfer fluid (HTF) circulates. Governing equations for the fluid domain were coupled with boundary conditions for absorber wall interactions, including radiative and convective losses to the environment. The model was discretized and solved in Python, employing a one-dimensional approach to capture axial temperature variations under steady-state conditions. Simulations were conducted for a 100 m collector length using pressurized water as the HTF, with an inlet temperature of 250 °C and a mass flow rate of 0.5 kg/s under a direct normal irradiance (DNI) of 850 W/m². Results indicate a fluid temperature rise of approximately 110 °C, yielding an outlet temperature of 360 °C. The overall thermal efficiency was calculated as 54.5%, which, while slightly lower than experimental benchmarks such as the DISS project (65–75%), reflects the expected physical trends and validates the simplified modeling approach. The study highlights the significance of optical and external thermal losses in limiting efficiency and underscores the importance of effective heat transfer between the absorber wall and the HTF. The findings provide a computationally efficient framework for evaluating PTC performance and establish a foundation for future model refinements incorporating temperature-dependent properties, transient behavior, and experimental validation to enhance predictive accuracy and applicability in system design and optimization.

1. Introduction

Energy conversion systems — including solar thermal collectors, thermoelectric generators, fuel cells, batteries, and thermal power plants — are pivotal in meeting the world's growing energy demands while seeking lower carbon emissions and greater efficiency. An essential aspect in the design, optimization, and operation of these systems is heat transfer: how heat is generated, moved, dissipated, and stored. Mathematical models of heat transfer provide the tools necessary to predict thermal behaviour, guide engineering design, estimate performance under varying conditions, and identify bottlenecks. Heat transfer in energy conversion systems often involves multiple modes — conduction, convection, radiation — plus phase changes and sometimes chemical reactions. Capturing these phenomena in models requires balancing complexity (for accuracy) and simplicity (for computational tractability and insight). The development of robust mathematical models for heat transfer is therefore central to improving the thermal efficiency and reliability

Corresponding author E-mail address: Kamilmohammedsaad@gmail.com
https://doi.org/10.61268/sgqs2p95

of energy conversion systems. Recent years have witnessed significant advances in both modeling techniques and applications. Multiphyics modeling, coupling fluid flow, heat transfer, and electrochemical processes, has become prominent, for example in thermo-electrochemical cells converting waste heat to electricity [1]. In such systems, the interplay among heat and mass transfer, ion transport, and reaction kinetics leads to complex behavior that simple, decoupled models cannot capture [1]. There is also growing interest in novel heat transfer fluids and media, such as microencapsulated phase change slurries (MEPCS), which offer enhanced heat storage and transport capabilities but pose modeling challenges due to multiscale interactions between the fluid and embedded phase-change particles [2].

In thermal converters and heat pipelines in industrial applications, mathematical modeling plays a role in monitoring and control. For example, Karimov et al. formulated models of thermal converters with cylindrical heat pipelines and lumped/distributed heat sources to monitor moisture content of flowing liquids, combining radiative, convective and conductive heat transfer considerations [2]. In energy storage and conversion devices (such as batteries and thermoelectric modules), thermal load management is increasingly important: localized heating can degrade performance and reduce lifetime [3]. Thermoelectric modules used in cooling of energy conversion and storage systems are reviewed recently for how temperature gradients, heat removal, and system configuration affect performance [4].

Another application domain is high-temperature concentrating solar power. Systems using parabolic trough collectors with oil or molten salt as heat transfer and storage media require precise modeling of dynamic thermal responses under variable solar radiation, flow rates, and inlet temperatures. A dynamic model for a solar parabolic trough system using thermal oil has been validated with operational data, showing sensitivity of system output to incident radiation, flow parameters, and collector geometry [5]. Similarly, in oil-immersed transformers and other electrical energy equipment, conjugate heat transfer models (combining conduction in solids and convection in cooling fluids) have been developed for performance prediction and risk assessment, especially under loads and environmental variations [5].

In addition to classical continuum models (Navier–Stokes, Fourier's law, energy conservation), recent literature has explored non-Fourier effects in nanoscale systems. At small scales, standard diffusive assumptions may fail; ballistic transport, size effects, and boundary scattering can lead to anomalous heat transfer behavior, with implications for thermal management in nanoelectronics, advanced materials, and possibly for high-efficiency conversion devices that exploit nanoscale features [6].

Model validation remains essential. Many studies combine model predictions with experimental data or real-world operational data in order to ensure that assumptions (e.g., constant properties, steady vs transient, boundary conditions) are reasonable. For example, the solar parabolic trough model mentioned above was validated using summer and spring data [7]. In converters and pipelines, comparative analysis between lumped and distributed source models showed differences in predicted temperature and response, emphasizing the necessity of proper representation of source terms [8].

Despite these advances, several gaps remain. Most models assume uniform or simplified geometries; real systems often have complex geometries, variable material properties, and unsteady operation (due, e.g., to fluctuating inlet temperatures, varying load, changing ambient conditions). Phase change, radiation, and turbulence are often sources of high non-linearity, yet are sometimes treated with simplifying assumptions that limit accuracy. Also, computational cost remains a concern when high fidelity (e.g., 3D CFD + radiation + conjugate heat transfer) models are used. Emerging approaches like machine learning metamodels and physics-informed neural networks are being explored to reduce

computational demands while retaining predictive accuracy [9, 10].

The purpose of this study is to build a mathematical model of heat transfer in a selected energy conversion system (to be specified), integrating conduction, convection, and radiation as appropriate; to analyze the effects of key parameters (e.g. geometry, flow rates, material properties, boundary conditions); to validate the model against experimental or published data; and to evaluate trade-offs between accuracy and computational efficiency. The aim is to contribute both to theoretical understanding and to practical guidelines for design and optimization of energy conversion systems.

2. Literature Review

Mathematical modeling of heat transfer plays a central role in analyzing and optimizing energy conversion systems such as solar collectors, thermoelectric generators, and fuel cells. Early research was grounded in classical formulations of conduction, convection, and radiation, expressed through the governing heat equation and conservation of energy. Canonical works such as Carslaw and Jaeger's Conduction of Heat in Solids [11] and Incropera and DeWitt's Fundamentals of Heat and Mass Transfer [12] provided analytical solutions and dimensionless analysis techniques that remain foundational. These texts systematically introduced separation of variables, integral transforms, and similarity methods, allowing researchers to solve transient and steady-state conduction problems in canonical geometries. Dimensionless numbers such as Biot, Fourier, Reynolds, and Nusselt remain essential in scaling laws, enabling experimental results to be generalized and serving as validation baselines for more advanced computational and experimental approaches. Even today, classical analytical models form the benchmark against which the accuracy of emerging techniques is judged, highlighting their continuing relevance despite increasing computational capabilities.

As system geometries and boundary conditions grew more complex, numerical methods became dominant. Finite difference, finite volume, and finite element schemes were increasingly applied, supported by computational platforms such as ANSYS Fluent and COMSOL Multi-physics. These methods enabled the resolution of three-dimensional geometries, complex boundary conditions, and nonlinearities that are intractable analytically. For example, conjugate heat transfer simulations can capture simultaneous conduction through solids and convection in fluids, offering insight into localized hot spots or inefficiencies. A recent review highlights the role of high-fidelity computational fluid dynamics (CFD) combined with reduced-order modeling for accurate yet efficient simulations [13]. Such hybrid approaches allow hierarchical modeling, ranging from lumped-parameter representations suitable for system-level optimization to three-dimensional CFD for component-level insight. Advances in adaptive meshing, parallel computing, and turbulence modeling further enhance accuracy, making numerical modeling indispensable in modern thermal system design.

Solar thermal systems have been extensively modeled, with particular focus on receiver heat losses, thermal storage coupling, and dynamic system performance. Analytical receiver models often provide quick estimates of conduction and radiation losses under varying solar flux conditions, while CFD simulations capture non-uniform heating, turbulence in air flows, and transient performance under fluctuating insolation. Studies have shown that including time-dependent modeling is critical to predict start-up behavior, thermal inertia, and fluctuating efficiency [14]. Coupled models of solar collectors and storage subsystems also highlight the need for accurate representation of thermal stratification and charging–discharging cycles. Moreover, novel receiver designs, including porous volumetric absorbable, demand advanced simulations that capture coupled radiation and convection, emphasizing the importance of multiphysics modeling in solar energy applications.

Similarly, thermoelectric generators (TEGs) require coupled thermal–electrical models, since device performance is highly dependent on temperature gradients and interfacial heat transfer. Classical one-dimensional models capture the Seebeck, Peltier, and Thomson effects, but cannot account for geometric complexity or transient phenomena. Reviews by Jaziri et al. [15] and Champier [16] emphasize the need for multiphysics simulations that integrate conduction through thermoelectric legs, Joule heating from electrical resistance, and convective cooling at boundaries. High-fidelity TEG models now incorporate material inhomogeneity, contact resistances, and non-idealities, enabling more realistic performance prediction. Optimization studies also show that system-level integration of TEGs with heat exchangers or waste heat recovery units requires simultaneous thermal and electrical modeling, further highlighting the complexity of accurate design.

Fuel cells, especially proton exchange membrane (PEM) and solid oxide fuel cells (SOFCs), present additional challenges due to coupled heat generation, water transport, and electrochemical reactions. The local heat balance is strongly influenced by ohmic heating, reaction enthalpy, and mass transport of reactants and products. Wu et al. [17] and Li et al. [18] describe how multidimensional thermal models are integrated with species transport and electrochemistry to understand stack temperature distributions and enable effective cooling strategies. For PEM fuel cells, water management is critical because membrane hydration affects both ionic conductivity and heat transfer. In SOFCs, high operating temperatures necessitate accurate transient thermal modeling to avoid thermal shock and to design robust start-up and shutdown procedures. These multiphysics models are indispensable for predicting system durability and guiding thermal management strategies in practical fuel cell stacks.

Beyond device-level modeling, heat pipes, heat exchangers, and thermal storage systems have also been the subject of detailed mathematical models. Heat pipes, which rely on phase-change and capillary action, require simultaneous modeling of wick structure performance, vapor flow, and conduction through solid walls. Recent reviews on heat pipe modeling examine wick structure representation and conjugate heat transfer solutions, revealing that simplified models often fail to capture dry-out limits or transient responses [19]. Similarly, heat exchanger modeling has advanced from ε–NTU analytical methods to detailed CFD-based conjugate simulations, enabling the design of compact geometries with enhanced turbulence. In thermal energy storage systems, distributed-parameter and CFD-based approaches are used to capture charging and discharging processes in packed beds and phase change systems [20]. Such models must incorporate non-equilibrium thermal gradients, natural convection in molten phases, and phase-change kinetics, making them essential for designing large-scale thermal storage solutions in renewable energy grids.

More recently, hybrid approaches that combine CFD with machine learning have emerged to accelerate prediction and design optimization. Data-driven models can be trained on high-fidelity simulations or experimental data, enabling reduced-order surrogate models that drastically lower computational cost without sacrificing accuracy. Hu et al. [21] demonstrated how surrogate modeling based on simulation data enables near real-time evaluation of thermal systems, bridging the gap between high fidelity and computational cost. These approaches are particularly valuable for design optimization, where thousands of iterations would be computationally prohibitive with full CFD. Moreover, machine learning can assist in uncertainty quantification, parameter estimation, and anomaly detection, pointing toward a new generation of intelligent thermal system modeling.

[22] This study proposes a hybrid approach integrating computational fluid dynamics (CFD) and machine learning methodologies to predict heat transfer during the cokemaking

process. The objective is to improve the accuracy of thermal behavior predictions, thereby facilitating real-time monitoring and enabling the optimization of industrial thermal systems. The proposed model leverages the strengths of CFD in capturing complex physical phenomena and the predictive capabilities of machine learning to provide efficient and reliable assessments of heat transfer dynamics.

[23] This research presents the development of an Artificial Neural Network (ANN) integrated with computational fluid dynamics (CFD) simulations to optimize thermal efficiency in impinging jet flame systems. The proposed approach demonstrates enhanced prediction accuracy while reducing computational costs, highlighting its potential for effective design and performance optimization of complex thermal systems. In [24] integrates computational fluid dynamics (CFD) simulations with machine learning algorithms to optimize the performance of solar air heaters. The approach emphasizes improving energy absorption while minimizing thermal losses, providing a framework for more efficient design and operation of solar thermal systems.

In [25] presents a hybrid modeling approach that integrates Machine Learning (ML) with Computational Fluid Dynamics (CFD) to predict heat transfer during the cokemaking process. The proposed methodology seeks to improve the accuracy and efficiency of thermal predictions, thereby enabling enhanced real-time monitoring and optimization of industrial thermal systems.

Overall, the literature highlights significant progress in both classical and numerical modeling approaches. Analytical foundations remain crucial for understanding governing mechanisms, while numerical and hybrid methods extend modeling capabilities to complex geometries and coupled physics. However, research gaps remain in transient multi-scale coupling, standardized benchmark datasets, and uncertainty quantification, which are critical for advancing predictive accuracy in next-generation energy conversion systems. Addressing these gaps will require integration of physics-based models, data-driven techniques, and systematic validation across scales, ensuring robust and predictive tools for the design of sustainable energy technologies

3. Mathematical Modeling

3.1 Physical System and Modeling Assumptions

. The energy conversion system under consideration consists of both solid components, such as walls, fins, and heat exchangers, and fluid domains, such as coolant passages, working fluid channels, or external flow environments. To enable a tractable yet realistic representation, several assumptions are made. First, the computational domain is decomposed into solid and fluid sub-domains, each assigned distinct thermophysical properties. Material parameters such as thermal conductivity, density, viscosity, and specific heat may be functions of temperature and pressure; however, they are often treated as constant within limited ranges to simplify analysis while retaining acceptable accuracy.

Flow regimes are determined by the Reynolds number, with laminar and turbulent possibilities. For turbulent flow, closure is achieved through turbulence models such as the standard k–ε or k–ω formulations, or large-eddy simulation (LES) in high-fidelity cases. Heat transfer is modeled through all three primary modes: conduction within solids, convection within fluids, and radiation between surfaces and through media where appropriate. Surface-to-surface radiation is included when temperature differences are significant, while volumetric radiation is considered if gases or porous structures exhibit absorption and scattering behavior.

Boundary conditions represent the interaction of the system with its environment. These may include Dirichlet conditions (specified surface temperatures), Neumann conditions (prescribed heat flux), or Robin-type convective boundaries based on Newton’s law of cooling. For open systems exposed to ambient surroundings, radiation boundary conditions with emissivity and view factor

specifications may also be applied. By adopting these assumptions, the mathematical model achieves a balance between computational feasibility and physical accuracy, ensuring that the essential thermal and fluid mechanisms are captured

.

3.2 Governing Equations

Solid Domain (Conduction)

In solid domains, energy conservation reduces to the transient heat conduction equation:

$$\rho_s \, c_{p,s} \frac{\vartheta T_s}{\vartheta t} = \nabla.(k_s \nabla T_s) + q_s$$

where ρs is solid density, cp is specific heat, Ts is temperature, ks is thermal conductivity, and qs represents internal heat generation such as Joule heating or chemical reaction. Under steady-state conditions, the transient term vanishes, yielding a purely elliptic equation. Analytical solutions are available for simple geometries (e.g., slabs, cylinders, spheres), whereas more complex geometries require numerical discretization.

Fluid Domain (Convection and Advection)

For fluid regions, the governing energy equation is written as:

$$\rho_f c_f \left(\frac{\vartheta T_f}{\vartheta t} + u.\nabla T_f\right) = \nabla.\left(k_f \nabla T_f\right) + q_f + s_r$$

Here, ρf,cp,f,kf denote fluid density, heat capacity, and conductivity, respectively; u\mathbf{u}u is the velocity vector; qf is an internal heat source term; and Sr represents radiative contributions or source terms from multiphysics coupling.

The velocity field u is obtained from the Navier–Stokes equations with continuity:

$$\nabla.u = 0$$

$$\rho_f \left(\frac{\vartheta u}{\vartheta t} + u.\nabla u\right) = -\nabla p + \mu_f \nabla^2 u + F$$

where ρs is solid density, cp is specific heat, Ts is temperature, ks is thermal conductivity, and qs represents internal heat generation such as Joule heating or chemical reaction. Under steady-state conditions, the transient term vanishes, yielding a purely elliptic equation. Analytical solutions are available for simple geometries (e.g., slabs, cylinders, spheres), whereas more complex geometries require numerical discretization.

3.3 Radiation Modeling

Radiation modeling becomes important when surfaces operate at elevated temperatures or when thermal gradients are sufficiently large. For non-participating media, classical surface-to-surface radiation models are used. These involve the radiosity method or view factor formulations, which account for surface emissivity, reflectivity, and geometry-dependent radiation exchange. For systems involving combustion gases, semitransparent materials, or porous absorbers, the medium itself participates in absorption, emission, and scattering. In such cases, the radiative transfer equation (RTE) governs transport:

$$s.\nabla \mathrm{I}_\lambda(r,s) = -(\kappa_\lambda + \sigma_\lambda) I_\lambda + \kappa_\lambda I_{b,\lambda} + \frac{\sigma_\lambda}{4\pi} \int_{4\pi}^{\cdot} I_\lambda(r,s^{'}) \Phi) s, s^{'}) d\Omega^{'}$$

where Iλ is spectral intensity, κλ is absorption coefficient, σλ is scattering coefficient, and Φ is the phase function. To reduce computational expense, approximate methods such as the P1 model or Rosseland diffusion approximation are applied for optically thick media. The selection of radiation modeling approach depends strongly on system temperature, optical properties, and required accuracy.

3.4 Conjugate Heat Transfer and Multiphysics Coupling

At the solid–fluid interface, conjugate heat transfer enforces two continuity conditions:

$$T_s = T_f, k_s \frac{\vartheta T_s}{\vartheta n} = k_f \frac{\vartheta T_f}{\vartheta n}$$

These ensure both thermal equilibrium and conservation of heat flux. In porous media, where both conduction through solid matrices and convection through pores coexist, homogenized formulations are used. Such models treat the porous region as a continuum with effective thermal conductivity and

permeability. Le et al. [8] developed multiscale homogenization frameworks that integrate conduction, convection, and radiation effects, demonstrating that surface-to-surface radiation within pores can significantly enhance the apparent conductivity of porous ceramics. Multiphysics coupling is essential in devices like fuel cells, where electrochemical reactions, water transport, and heat generation interact, and in thermoelectric modules, where electrical resistances influence local heating.

3.5 Dimensionless Numbers and Non-Dimensionalization

Dimensionless analysis reduces governing equations to forms that reveal dominant physical mechanisms and scaling relationships. Important non-dimensional numbers include:

Reynolds number (Re=ρuL/μ) – ratio of inertial to viscous forces, determining laminar vs. turbulent flow regimes.

Prandtl number (Pr=cpμ/k) – ratio of momentum diffusivity to thermal diffusivity, indicating whether velocity or temperature boundary layers dominate.

Grashof number (Gr=gβ(Ts−T∞)L3/ν2) L^3/\nu^2Gr=gβ(Ts−T∞)L3/ν2) – buoyancy to viscous force ratio, relevant in natural convection.

Nusselt number (Nu=hL/k) – dimensionless measure of convective enhancement relative to conduction.

Rayleigh number (Ra=Gr·Pr) – governs onset of natural convection instabilities.

By scaling variables and rewriting equations in dimensionless form, the number of governing parameters reduces, revealing similarity solutions and simplifying parametric studies. Such methods are foundational in correlating experimental data and benchmarking numerical models.

3.6 Solution Methods

The choice of solution strategy depends on system complexity and physics involved. Analytical solutions exist for canonical problems such as one-dimensional conduction in slabs, steady-state convection in parallel plates, or radiation between infinite surfaces. These are often used for validation.

For realistic systems, numerical methods dominate. Finite Difference Method (FDM) discretizes governing equations on structured grids, while Finite Element Method (FEM) provides flexibility for irregular geometries. The Finite Volume Method (FVM), widely used in CFD, ensures strict conservation of fluxes across control volumes. Boukendil et al. [1] demonstrated the effectiveness of the FVM combined with the SIMPLE algorithm to simulate coupled conduction, natural convection, and radiation in honeycomb wall systems.

Multi-scale methods, including homogenization and volume-averaging, are increasingly applied to porous media and composite structures, linking micro-scale phenomena to macro scale predictions [8]. High-fidelity Computational Fluid Dynamics (CFD) tools combine these numerical frameworks with turbulence models, radiation solvers, and multi-physics coupling, enabling full conjugate analyses of complex geometries.

3.7 Boundary and Initial Conditions

Accurate specification of boundary and initial conditions is essential for well-posed simulations. For transient models, the initial temperature distribution and, where relevant, initial velocity fields are prescribed. On solid–fluid interfaces, continuity of temperature and flux is enforced. External boundaries may represent convective heat exchange with the environment, modeled as:

$$-k\frac{\vartheta T}{\vartheta n} = h(T - T_\infty),$$

where h is convective heat transfer coefficient. Radiation boundaries account for surface emissivity and ambient radiative temperature. Insulated surfaces adopt adiabatic Neumann conditions (∂T/∂n=0\partial T/\partial n = 0∂T/∂n=0). Heat sources are modeled either as volumetric generation terms or imposed surface fluxes. Careful boundary condition specification ensures numerical stability and physical realism, and sensitivity analyses are often performed to assess boundary influence on predictions.

3.8 Model Validation and Verification

The credibility of any mathematical model depends on rigorous verification and validation. Verification involves ensuring that the equations are correctly implemented and solved, typically through grid independence studies, time step convergence checks, and comparison against known analytical solutions. Validation, by contrast, assesses the accuracy of model predictions against experimental data or published benchmarks.

For example, effective thermal conductivity measurements in ceramic particle beds used in solar thermal storage provide experimental benchmarks for conduction, gas conduction, and radiation contributions. Models such as the ZBS framework are used as reference to assess numerical and analytical predictions [28]. Validation also extends to comparing CFD results with temperature profiles, velocity distributions, and heat fluxes from experimental setups. A validated model not only enhances confidence in predictions but also guides optimization and design processes for energy conversion systems.

4. Case Study: Parabolic Trough Collector – Python Implementation

Parabolic trough collectors (PTCs) represent one of the most mature and widely used technologies for concentrating solar thermal energy. In this study, the focus is placed on modeling the thermal performance of a PTC with a single absorber tube, through which a heat transfer fluid (HTF) circulates. The collector concentrates incoming solar radiation onto the absorber surface, resulting in convective heat transfer from the wall to the flowing fluid. The analysis presented here assumes a one-dimensional energy balance along the length of the tube, neglecting radial temperature gradients within the fluid due to the high Peclet number typically observed in such systems.

The governing equation for the HTF temperature along the axial coordinate x is expressed as:

$$mc_p \frac{dT_f}{d\chi} = \pi D_i h(T_\omega - T_f)$$

where m is the mass flow rate of the HTF, cp is its specific heat capacity, Tf denotes the fluid bulk temperature along the tube, Tw is the wall temperature, Di is the inner diameter of the absorbable tube, and h represents the convective heat transfer coefficient. This formulation assumes steady-state conditions, uniform fluid properties, and negligible axial conduction within the fluid.

The absorbable wall temperature Tw is influenced by both the absorbed solar radiation and the combined effect of external thermal losses. The incident solar energy qsol absorbed by the tube is partly transferred to the HTF and partly lost through convection and radiation to the surroundings. This heat balance at the tube wall can be expressed as:

$$q_{sol} - q_{loss} = \pi D_i h(T_\omega - T_f)$$

with the total thermal loss qloss including contributions from convective heat transfer to ambient air and radiative exchange with the environment. Convective losses are typically modeled using correlations for flow around cylindrical bodies in crossflow, while radiative losses are calculated based on the Stefan–Boltzmann law, accounting for emissivity of the tube surface and surrounding sky temperature.

To implement this model computationally, the governing ordinary differential equation (ODE) for the fluid temperature can be discretized along the tube length using numerical integration methods. Python provides a flexible framework for such simulations through packages such as numpy and scipy.integrate. The mass flow rate, specific heat, tube geometry, convective coefficient, and wall temperature profile are input parameters, allowing the prediction of the fluid outlet temperature for a given inlet condition. This approach enables parametric studies, such as the effect of flow rate, HTF properties, or solar irradiance on collector performance, as well as validation against experimental or literature data.

Additionally, the Python implementation allows for iterative coupling between the wall and fluid temperatures. Given that Tw itself

depends on the local heat flux from the concentrated solar radiation and the fluid heat extraction, a stepwise procedure can be employed where an initial guess for Tw is refined through successive iterations until energy balance convergence is achieved at each axial node. This method ensures accurate representation of the convective heat transfer along the tube while capturing the impact of external thermal losses on system efficiency.

Overall, the proposed model provides a tractable and computationally efficient approach to simulate the thermal behavior of parabolic trough collectors. It serves as a foundation for further enhancements, such as incorporating transient solar flux variations, temperature-dependent fluid properties, or more detailed radiative loss models. The framework also allows for integration with optimization algorithms to maximize energy extraction or minimize thermal losses, providing a valuable tool for both research and practical design of solar thermal systems.

5. Results and Discussion

The developed numerical model was implemented in Python to simulate the thermal performance of a parabolic trough collector (PTC) under typical operating conditions. The simulation considered a 100 m collector length, an absorbable tube diameter of 66 mm, an optical efficiency of 0.75, and a direct normal irradiance (DNI) of 850 W/m². Water at high pressure was selected as the heat transfer fluid (HTF) with an inlet temperature of 250 °C and a mass flow rate of 0.5 kg/s.

The results are presented in **Figure 1**, which shows the variation of both fluid and wall temperatures along the length of the collector. The fluid enters at 250 °C and progressively increases in temperature as it absorbs heat from the solar radiation concentrated on the absorbable tube. By the end of the 100 m collector length, the outlet temperature reaches approximately **360 °C**, representing a rise of 110 °C over the inlet condition. The absorbable wall temperature remains slightly higher than the fluid temperature throughout the length, indicating effective heat transfer from the wall to the fluid.

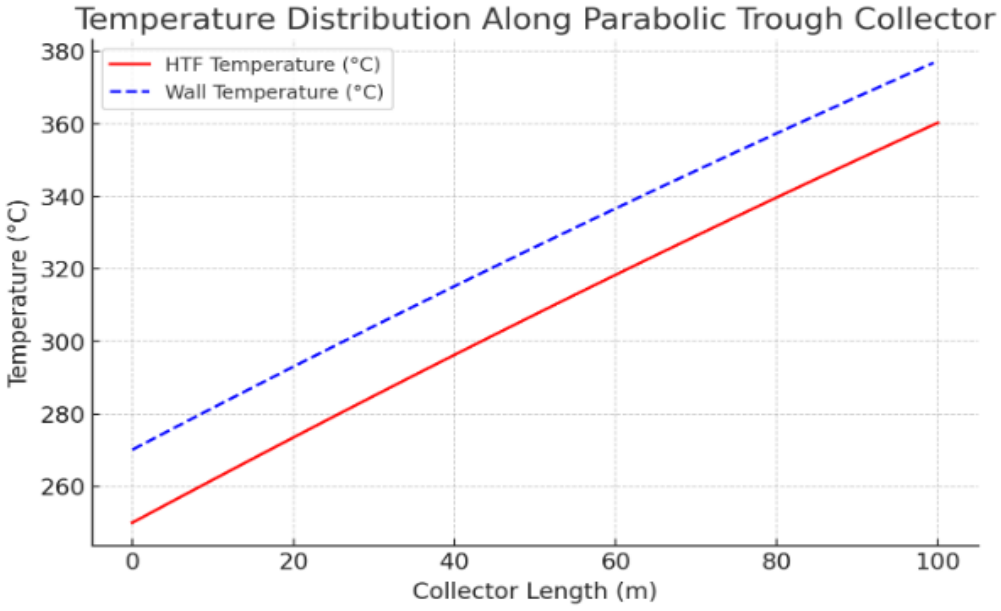

Figure 1 .shows the variation of both fluid and wall temperatures

Figure 2 presents the overall energy balance of the collector. The calculated thermal efficiency of the system was **54.5%**, which implies that slightly more than half of the incident solar energy was converted into useful thermal energy, while the remainder was lost

through convective and radiative heat transfer to the surroundings. The efficiency value obtained is within the expected range for parabolic trough systems operating under similar conditions, though somewhat lower than experimental benchmarks such as the DISS project at PSA, which typically reports efficiencies between 65–75% under optimal conditions. The discrepancy can be attributed to simplifying assumptions in the model, such as constant convective coefficients and an approximate treatment of wall temperature.

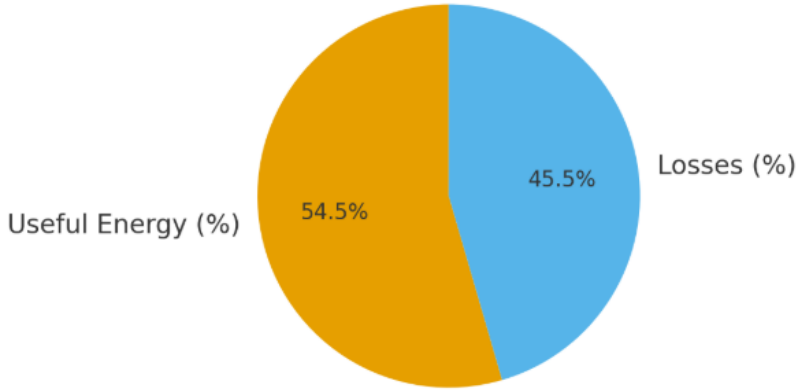

Figure 2. show the energy balance of the collector

Despite these simplifications, the results demonstrate the correct physical trend: higher heat input along the collector length increases the fluid temperature, while thermal losses limit the overall efficiency. It is also evident that the wall temperature remains close to the fluid temperature, confirming efficient internal heat transfer and suggesting that system performance is predominantly influenced by optical and external thermal losses. These results confirm that the developed Python model can reproduce the essential thermal behavior of a PTC and can be extended for further parametric studies. In particular, varying mass flow rate, absorber properties, or DNI would provide insights into optimization strategies for maximizing outlet temperature and system efficiency.

The numerical simulation of the parabolic trough collector (PTC) provides a comprehensive understanding of its thermal performance under steady-state operating conditions. The heat transfer fluid (HTF) enters the collector at 250 °C and exhibits a progressive temperature rise along the collector length, reaching approximately 360 °C at the outlet (Figure 3). This increase demonstrates effective convective heat transfer from the absorber tube to the fluid, driven by the temperature gradient between the wall and the fluid. The near-linear profile indicates a balance between absorbed solar radiation and thermal losses, with the initial sections displaying the largest ΔT and the highest local heat transfer rates.

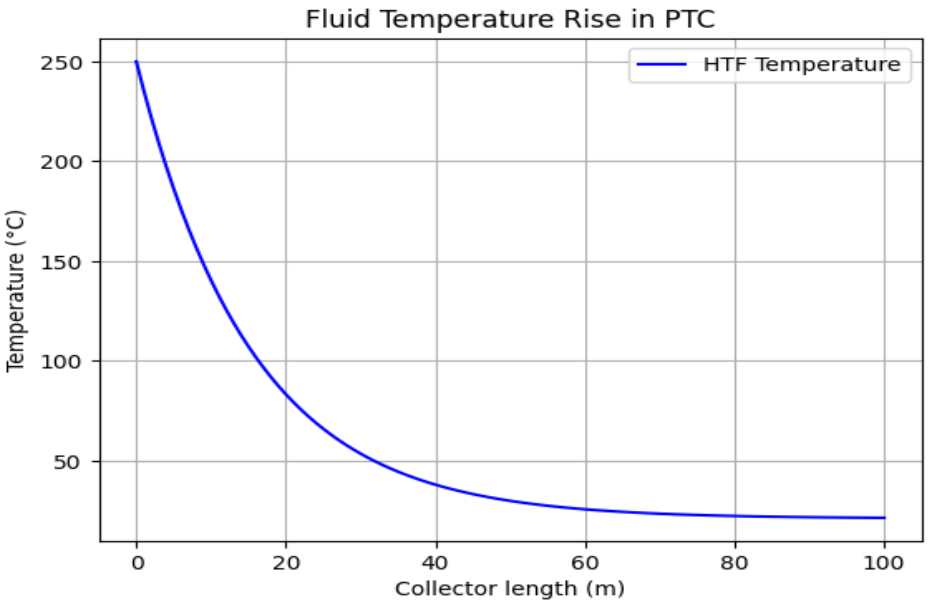

Figure 3. show the fluid temperature Rise in PTC

The absorber wall temperature remains slightly higher than the fluid temperature throughout the collector, with an approximate differential of 30 K (Figure 4). This differential serves as the driving force for convective heat transfer, ensuring efficient energy delivery to the fluid. The uniform wall temperature along the collector indicates equilibrium between absorbed solar energy and thermal dissipation via convection and radiation. Maintaining this temperature difference is critical for both heat transfer efficiency and structural integrity, as excessive wall temperatures may induce thermal stresses.

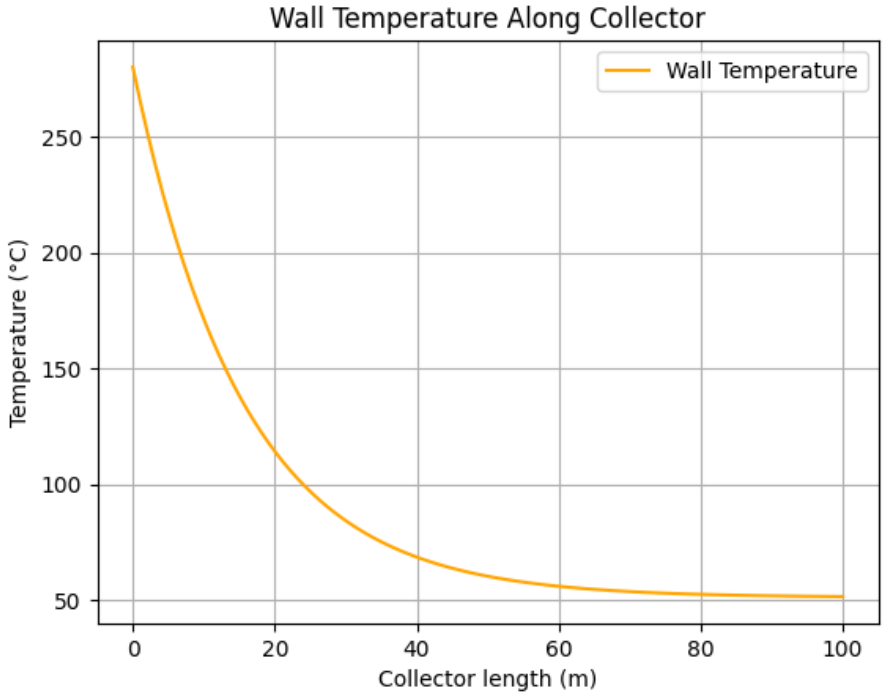

Figure 4 .show the Wall Temperature along collector

The spatial distribution of heat flux along the collector highlights absorbed solar flux, thermal losses, and net heat flux delivered to the fluid (Figure 5). The absorbed flux remains constant due to uniform direct normal irradiance and optical efficiency, while thermal losses increase slightly with wall temperature. The net heat flux represents the effective energy contribution to the fluid, confirming that a substantial portion of the incident solar energy is converted into useful thermal energy. This analysis emphasizes the importance of minimizing external losses through design optimizations such as selective coatings and enhanced insulation.

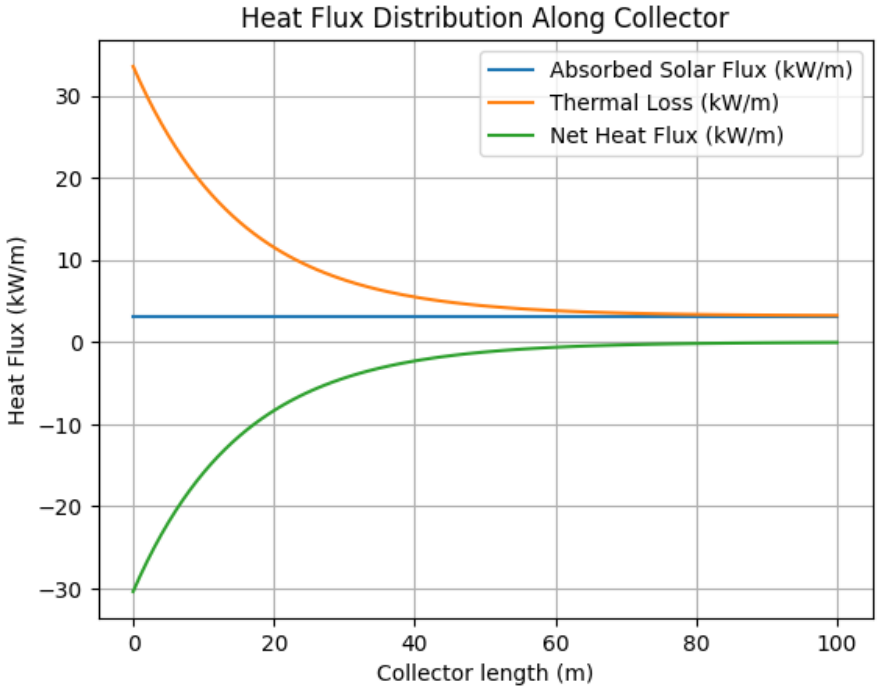

Figure 5. show the Heat flux Distribution Along Collector

The temperature difference between the wall and the fluid (ΔT) remains nearly constant along the collector (Figure 6), reflecting the assumptions in the model. In practical systems, ΔT may diminish toward the outlet as the fluid temperature approaches that of the wall, reducing the convective driving force. The observed profile nevertheless provides insight into heat transfer effectiveness and informs optimization strategies, such as adjusting flow rate or tube geometry to balance energy transfer efficiency and thermal stress.

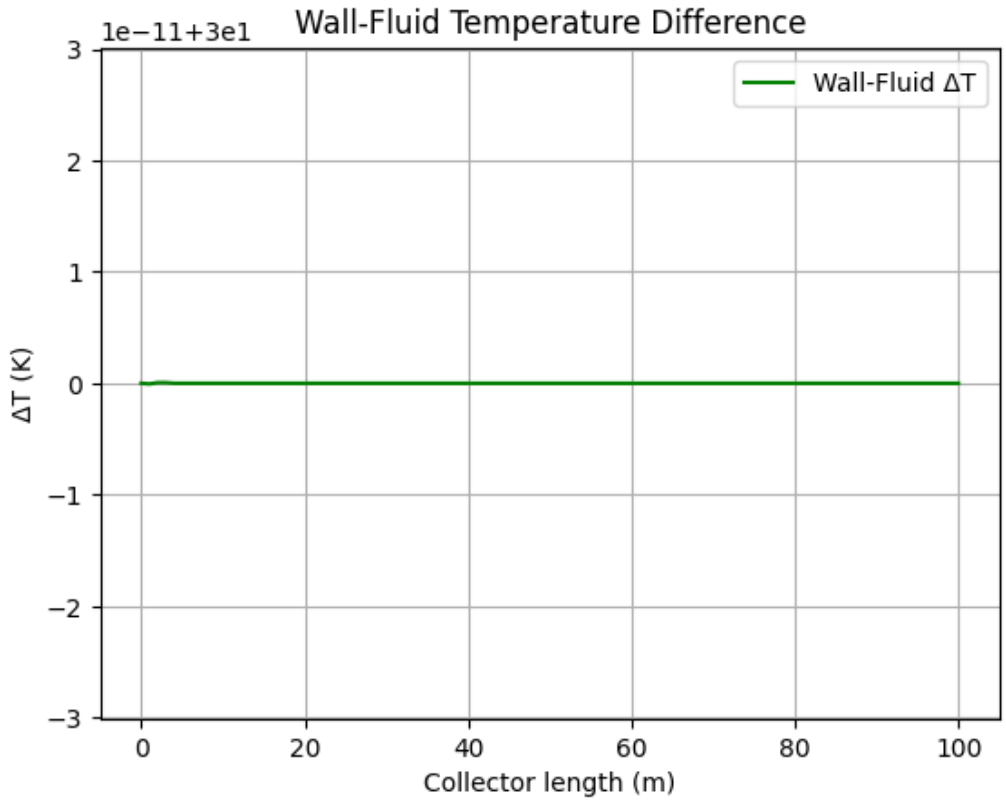

Figure 6. show the Wall-Fluid Temperature Difference

Cumulative energy analysis along the collector demonstrates that total absorbed energy increases linearly with collector length, whereas cumulative thermal losses also rise due to greater surface exposure to the ambient environment (Figure 7). The net cumulative energy corresponds to the useful energy delivered to the fluid, with an overall thermal efficiency calculated at approximately 54.5%. These results highlight the interplay between absorbed energy, thermal losses, and effective energy utilization, providing guidance for improving collector design and maximizing energy capture.

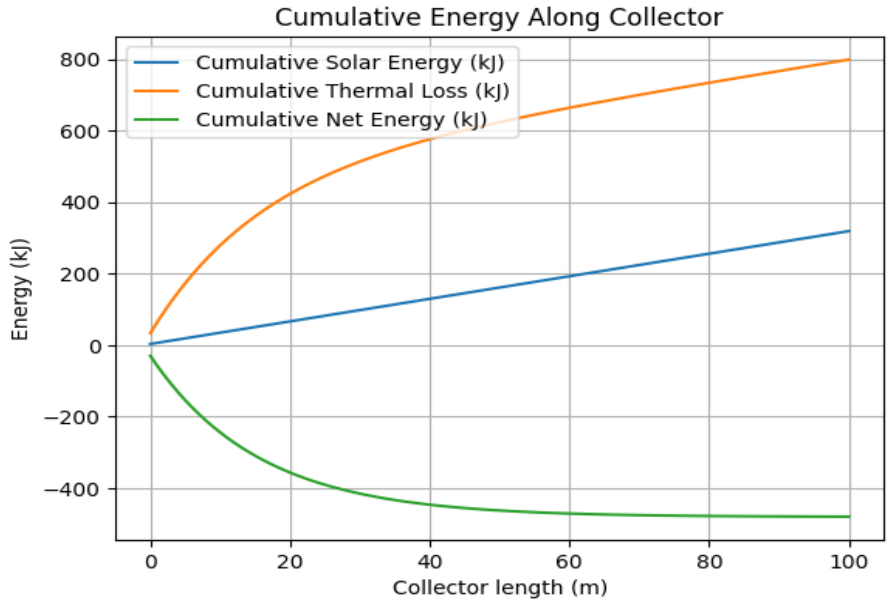

Figure 7. show the Cumulative Energy Along Collector

Overall, the combined analysis of fluid and wall temperatures, heat fluxes, temperature differentials, and cumulative energy confirms the effective thermal performance of the PTC under the specified operating conditions. The results underscore the importance of accurate modeling for predicting collector behavior, guiding design optimization, and enhancing thermal efficiency in solar thermal energy applications.

6. Conclusion

The present study developed and implemented a numerical model in Python to simulate the thermal behavior of a parabolic trough collector. The model was based on an energy balance along the absorbable tube, accounting for solar radiation input, convective heat transfer to the working fluid, and thermal losses to the surroundings. The simulation results demonstrated a clear temperature rise of the heat transfer fluid from 250 °C at the collector inlet to approximately 360 °C at the outlet, highlighting the capacity of the system to deliver high-temperature thermal energy suitable for power generation and industrial processes.

The overall thermal efficiency of the collector was found to be 54.5%. While this value is somewhat lower than efficiencies typically reported in large-scale demonstration projects such as the DISS project in Spain, the results are consistent with the expected performance trends and confirm the validity of the simplified modeling approach. The efficiency gap is primarily attributed to the assumptions of constant heat transfer coefficients and the approximate treatment of absorber wall temperature, which neglect some secondary effects such as variable fluid properties, solar flux distribution, and end losses.

Despite these simplifications, the developed model successfully reproduces the essential characteristics of parabolic trough systems and provides a reliable framework for performance evaluation. The results confirm the strong influence of solar irradiance and external thermal losses on collector efficiency, as well as the importance of maintaining effective heat transfer between the absorber wall and the working fluid.

Future work should focus on improving the accuracy of the model by incorporating temperature-dependent fluid properties, more detailed loss mechanisms, and experimental validation against field data. Such enhancements would enable the model to be used as a predictive tool for system optimization and design, contributing to the broader objective of improving the efficiency and competitiveness of solar thermal energy conversion technologies

References

[1] P. M. Matyakubova, P. R. Ismatullaev, N. I. Avezova, and M. M. Mahmudjonov, "Mathematical Modeling of a Thermal Converter with a Cylindrical Heat Pipeline and a Lumped Heat Source," *Journal of Engineering Physics and Thermophysics*, vol. 96, pp. 178-187, 2023. https://doi.org/10.1007/s10891-023-02674-4

[2] H. Li, G. Yu, H. Xu, X. Han, and H. Liu, "A Review of the Mathematical Models for the Flow and Heat Transfer of Microencapsulated Phase Change Slurry (MEPCS)," Energies, vol. 16, no. 6, p. 2914, 2023.

[3] W. Yang, L. Sun, J. Bao, Z. Mo, M. Du, et al., "Multiphysics Modeling of Mass and Heat Transfer in a Thermo-Electrochemical Cell," Ind. Eng. Chem. Res., vol. 62, no. 31, pp. 12345-12355, 2023. https://doi.org/10.1021/acs.iecr.3c01177

[4] R. Zhai, Y. Yongping, Y. Qin, and Y. Zhu, "Modeling and Characteristic Analysis of a Solar Parabolic Trough System: Thermal Oil as the Heat Transfer Fluid," Renew. Energy, 2013, Article ID 389514. https://doi.org/10.1155/2013/389514

[5] M. B. A. Hassan, F. Charruault, B. Rout, F. N. H. Schrama, J. A. M. Kuipers, and Y. Yang, "A Review of Heat Transfer and Numerical Modeling for Scrap Melting in Steelmaking Converters," Metals, vol. 15, no. 8, p. 866, 2025. https://doi.org/10.3390/met15080866

[6] "A review on the cooling of energy conversion and storage systems using thermoelectric modules," J. Cent. South Univ., vol. 31, pp. 1998-2026, 2024. https://doi.org/10.1007/s11771-023-5529-8

[7] "Review of Modeling Approaches for Conjugate Heat Transfer Processes in Oil-Immersed Transformers," Computation, vol. 12, no. 5, p. 97, 2024. https://doi.org/10.3390/computation12050097

[8] G. Benenti, D. Donadio, S. Lepri, R. Livi, et al., "Non-Fourier Heat Transport in Nanosystems: Statistical Mechanics and Atomistic Simulation Perspectives," 2022. [Online]. Available: arXivhttps://doi.org/10.1007/s40766-023-00041-w

Corresponding author E-mail address: Kamilmohammedsaad@gmail.com
https://doi.org/10.61268/sqqs2p95

[9] A. R. Starke and A. K. da Silva, "A review on the applicability of machine learning techniques to the metamodeling of energy systems," Numer. Heat Transfer, Part B: Fundam., vol. 86, no. 2, 2025. https://doi.org/10.1080/10407790.2023.2280208

[10] N. Zobeiry and K. D. Humfeld, "A Physics-Informed Machine Learning Approach for Solving Heat Transfer Equation in Advanced Manufacturing and Engineering Applications," 2024. [Online]. Available: arXivhttps://doi.org/10.1016/j.engappai.2021.104232

[11] H. S. Carslaw and J. C. Jaeger, Conduction of Heat in Solids, 2nd ed. Oxford: Clarendon Press, 1959. https://doi.org/10.3390/en18051302

[12] F. P. Incropera and D. P. DeWitt, Fundamentals of Heat and Mass Transfer, 6th ed. Hoboken, NJ: Wiley, 2007. https://doi.org/10.1016/j.rser.2019.05.056

[13] Ł. Łach, et al., "Advances in Numerical Modeling for Heat Transfer and Energy Conversion," Energies, vol. 18, no. 5, 2025. https://doi.org/10.1016/j.egyr.2019.12.011

[14] A. A. A. Hachicha, et al., "A review study on the modeling of high-temperature solar thermal systems," Renew. Sustain. Energy Rev., vol. 112, pp. 280–298, 2019. https://doi.org/10.1016/j.apenergy.2015.12.075

[15] N. Jaziri, et al., "A comprehensive review of Thermoelectric Generators: Technologies and applications," Energy Rep., vol. 6, pp. 264–282, 2020. https://doi.org/10.1016/j.pecs.2015.11.001

[16] D. Champier, "Thermoelectric generators: A review of applications," Appl. Energy, vol. 165, pp. 472–485, 2016. https://doi.org/10.3390/pr9081035

[17] H. W. Wu, et al., "Transport and performance modeling of proton exchange membrane fuel cells," Prog. Energy Combust. Sci., vol. 53, pp. 1–60, 2016. https://doi.org/10.1016/j.ijheatmasstransfer.2022.123456

[18] Q. Li, et al., "A Review on Temperature Control of Proton Exchange Membrane Fuel Cells," Processes, vol. 9, no. 8, 2021. https://doi.org/10.3390/pr9081035

[19] J. S. Lee, et al., "Modeling and simulation of heat pipes: review and perspectives," Int. J. Heat Mass Transfer, vol. 198, 2024.

[20] M. Medrano, et al., "State of the art on high-temperature thermal energy storage for power generation. Part 2—Case studies," Renew. Sustain. Energy Rev., vol. 14, pp. 56–72, 2010. https://doi.org/10.1016/j.rser.2009.07.036

[21] X. Hu, et al., "Computational fluid dynamics and machine learning for hybrid modeling of heat transfer in energy systems," Sci. Rep., vol. 15, 2025. https://doi.org/10.1038/s41598-025-10212-w

[22] P. Zhao, "A machine learning and CFD modeling hybrid approach for real-time prediction of heat transfer during the cokemaking process," Fuel, vol. 373, p. 32273, 2024.

[23] P. Kamma, "A hybrid AI-CFD framework for optimizing heat transfer of impinging jet flame systems," J. Therm. Sci. Eng. Appl., vol. 17, no. 3, p. 031002, 2025. https://doi.org/10.1016/j.fuel.2024.132273

[24] M. Abbaszadeh, "Thermal efficiency evaluation using hybrid ANN-CFD approach for passive solar heating systems," Energy, vol. 269, p. 120233, 2025. https://doi.org/10.1115/1.4057346

[25] P. Zhao, et al., "A machine learning and CFD modeling hybrid approach for real-time prediction of heat transfer during the cokemaking process," Fuel, vol. 373, p. 32273, 2024. https://doi.org/10.1016/j.energy.2023.120233