Future Technology
Open Access Journal
ISSN 2832-0379

November 2026| Volume 05 | Issue 04 | Pages 133-145

Journal homepage: https://fupubco.com/futech

https://doi.org/10.55670/fpll.futech.5.4.11

Future Publishing LLC

Review

Advances in computational fluid dynamics for enhanced heat transfer applications in engineering systems: a critical review and comparative synthesis

Manjula G. J.[1]*, Abhijeet Prakash Shah[2], Rajaram M. Shinde[3], P S V Ramana Rao[4], Avinash H. Kolekar[5], Mohammed Hameeduddin Haqqani[6]

[1]Department of Mathematics, Siddaganga Institute of Technology, Tumakuru – 572103, India
[2]Mechanical Engineering Department, Kasegaon Education Society's Rajarambapu Institute of Technology, affiliated to Shivaji University, Sakhrale, MS- 415414, India
[3]Department of Robotics and Automation, Kasegaon Education Society's Rajarambapu Institute of Technology, affiliated to Shivaji University, Sakhrale, MS-415414, India
[4]Department of Mechanical Engineering, Centurion University of Technology and Management, Vizianagaram, Andhra Pradesh India, India
[5]Vidya Prathishthan's Kamalnayan Bajaj Institute of Engineering and Technology Baramati: Pune, Maharashtra, 413133, India
[6]Department of Mechanical Engineering, Career Point University, Kota, Rajasthan, Pin code- 325003, India

ARTICLE INFO

Article history:
Received 22 February 2026
Received in revised form
12 July 2026
Accepted 17 August 2026

Keywords:
Computational fluid dynamics,
Heat transfer enhancement, Heat exchangers, Nanofluids, Thermal–hydraulic performance, Topology optimization

*Corresponding author
Email address:
gjm@sit.ac.in

DOI: 10.55670/fpll.futech.5.4.11

ABSTRACT

Computational Fluid Dynamics has become an important tool for evaluating heat-transfer enhancement in engineering systems where thermal performance must be balanced against pressure loss, energy consumption, and design constraints. This critical review and comparative synthesis examine published CFD investigations of selected heat-exchange applications, including shell-and-tube, double-pipe, twisted-tube, and ground-air heat exchangers. The reviewed studies include nanofluid-based enhancement, geometric modification, internal turbulators, and optimization-based design strategies. The selected literature is compared in terms of evidence for heat-transfer improvement, hydraulic penalties, numerical modeling options, validation methods, and practical applications. The synthesis shows that geometric changes and passive flow-disruption methods can yield significant thermal gains, but at the cost of higher-pressure losses. Nanofluids are also a potential means of further improving heat transfer with relatively minor structural modification, but their long-term stability, fouling characteristics, and pumping needs are still practical issues to be addressed. The comparison also shows that no single enhancement method is always best; performance depends on the exchanger configuration, operating conditions, flow regime, and design objectives. The review highlights the need for consistent thermal–hydraulic performance assessment, better experimental validation, and more attention to manufacturability and long-term operation. It concludes with application-oriented guidance and future priorities for reliable, energy-efficient CFD-assisted thermal-system design.

1. Introduction

Effective heat transfer underpins the performance and reliability of engineering systems in energy conversion, process industries, heating, ventilation, and air conditioning, electronics cooling, aerospace, and thermal management. Enhancement solutions that boost heat-transfer rates without excessive pressure loss, pumping power, or material cost are more crucial than ever because of the need for energy savings, system size restrictions, and rising thermal loads. Because it enables analysis of fluid flow behavior, temperature distribution, boundary-layer development, and solid-fluid interactions across a range of geometrical and operating parameters, computational fluid dynamics (CFD) has emerged as a crucial analytical tool in this discipline. For instance, numerical study of Y-shaped inserts with perforations has shown that passive flow-disruption devices can enhance heat transfer through secondary flow and turbulence generation, which may result in increased pressure loss [1].

CFD has also facilitated the investigation of conjugate and Multiphysics heat-transfer processes that are difficult to evaluate through experimentation alone. Numerical modeling of industrial anode-baking furnaces has demonstrated the importance of simultaneously resolving conduction through solid components and convection within fluid domains [2]. Similarly, CFD investigations of open-cell foam catalytic reactors have shown that radiative heat transfer must be considered to avoid inaccurate prediction of temperature distributions and reaction behavior [3]. These studies demonstrate that reliable thermal analysis requires appropriate representation of the physical mechanisms governing each engineering application. At smaller scales, experimental and computational investigations have improved understanding of forced convection in microchannels and identified the sensitivity of thermal performance to channel dimensions, fluid properties, and operating conditions [4]. More recent studies of leaf-vein-inspired microchannel heat sinks and V-shaped wavy microchannels have reported improved temperature uniformity, reduced thermal resistance, and intensified fluid mixing through geometrical modification [5,6]. These microscale studies provide relevant theoretical context for CFD-based heat-transfer enhancement. However, the comparative evidence synthesis in this review is limited to selected shell-and-tube, double-pipe, twisted-tube, and ground-air heat-exchanger applications; it does not include a dedicated microchannel case study.

Optimization-driven CFD methods have further expanded the role of numerical modeling from performance evaluation to design exploration. Multi-objective topology optimization can simultaneously address thermal resistance and pressure loss, producing flow paths and solid structures that conventional parametric design may not identify [7]. CFD coupled with surrogate modeling has also been applied to predict and optimize heat-transfer behavior in aeroengine turbine casings, demonstrating the potential of reduced-order approaches for computationally demanding thermal systems [8]. Furthermore, CFD-based optimization of micro-baffles for film-boiling applications shows that geometric modification can increase heat-transfer stability under high-heat-flux conditions [9]. These advances reflect the growing relevance of optimization, surrogate modeling, and geometry-driven design in thermal engineering. However, the published literature remains highly disjointed in enhancement strategies, system configurations, flow regimes, and performance measures. Many studies focus on a particular geometry, working fluid, or application and are therefore not readily comparable between systems. Validation procedures, mesh-independence studies, turbulence-model choices, and the presentation of thermal–hydraulic trade-offs also differ significantly. Moreover, enhancements in heat-transfer rate are not always considered alongside pressure-drop penalties, pumping-power requirements, manufacturability, nanofluid stability, and long-term operational constraints. Therefore, a critical synthesis is needed to understand the performance of various enhancement techniques under different engineering conditions and to identify limitations to their practical implementation. Therefore, this research critically evaluates and summarizes published CFD studies of heat-transfer enhancement in common engineering systems in a comparative manner. Five specific applications of CuO–water nanofluids in shell-and-tube heat exchangers, shell-and-tube optimization using CFD, TiO_2–water nanofluids in double-pipe heat exchangers, twisted-tube heat exchangers using different nanofluids, and internal turbulators in ground-air heat exchangers are the main topics of the review.

The novelty of the article lies in the structured synthesis of existing evidence, the comparison of thermal–hydraulic trade-offs, the review of numerical validation practices, and the engineering guidance for applications. It does not include new CFD simulations, experimental data, or author-led model validation. The aim of this critical review is to:

- To synthesize published CFD approaches for enhancing heat transfer across selected engineering systems.
- To compare the thermal improvements and hydraulic penalties associated with nanofluids, geometric modification, passive flow-disruption techniques, and optimization-driven designs.
- To evaluate the numerical-modeling assumptions, validation practices, practical applicability, and limitations reported in the selected studies.
- To formulate application-oriented design guidance and identify priorities for future CFD-assisted heat-transfer research.

2. Theoretical background

2.1 Heat transfer mechanisms in engineering systems

Heat transfer in engineering systems occurs through conduction, convection, and radiation, either independently or through coupled mechanisms. In the five heat-exchanger applications reviewed in this article, forced convection governs energy transport within the working fluids, whereas conduction through tube walls and other solid components enables heat exchange between separated fluid streams. Accurate representation of this solid–fluid interaction is essential in conjugate heat-transfer modeling because errors in material properties, interface conditions, or near-wall resolution can directly affect the predicted temperature and heat-flux distributions [10,11]. Radiation is not a dominant mechanism in the five selected cases but remains relevant to the broader application of CFD in high-temperature thermal systems. Most heat-transfer enhancement techniques operate by modifying the thermal boundary layer, increasing fluid mixing, enlarging the effective heat-transfer area, or improving the thermophysical properties of the working fluid. Turbulence and secondary-flow development can increase convective transport by renewing the fluid adjacent to the heated surface. However, intensified mixing generally increases wall shear stress and pressure loss. Consequently, assess enhancement performance using both thermal and hydraulic indicators, not just heat-transfer improvement.

Table 1 summarizes the main heat-transfer mechanisms and their applicability to the chosen applications. Nanofluid-based cases mainly alter the thermal characteristics of the working fluid, while the optimized shell-and-tube and twisted-tube cases induce flow redistribution and secondary motion. In the ground-air heat exchanger, internal turbulators are intentionally used to disrupt the boundary-layer development. Conjugate heat transfer is still present in all 5 cases because energy is transferred through solid boundaries between interacting thermal domains.

Table 1. Heat-transfer mechanisms and their relevance to the reviewed CFD cases

Heat-transfer aspect	Physical basis	CFD-modelling requirement	Relevance to the reviewed cases
Conduction	Molecular energy transfer through solid walls, tubes, fins, and stationary media	Accurate solid-domain geometry, mesh resolution, and temperature-dependent material properties	Governs heat transfer through the separating walls in the shell-and-tube, double-pipe, twisted-tube, and ground-air systems
Forced convection	Energy transport produced by fluid motion over or through a heated surface	Accurate velocity field, inlet conditions, fluid properties, and thermal boundary conditions	Represents the principal heat-transfer mechanism in all five selected cases
Radiation	Thermal-energy exchange through electromagnetic emission and absorption	Appropriate surface or participating-medium radiation model and radiative properties	Provides broader high-temperature CFD context but is not the principal mechanism in the selected cases
Thermal-boundary-layer development	Formation of velocity and temperature gradients near solid surfaces	Refined near-wall mesh and suitable wall-treatment strategy	Directly affected by twisted tubes, optimized flow passages, and internal turbulators
Turbulence and secondary-flow mixing	Enhanced momentum and energy transport caused by fluctuations, vortices, and cross-stream motion	Appropriate turbulence model, near-wall treatment, convergence criteria, and grid-independence assessment	Central to the shell-and-tube optimization, twisted-tube, and turbulator-based cases
Conjugate heat transfer	Simultaneous conduction through solids and convection within adjacent fluids	Coupled solution of solid and fluid energy equations with consistent interface conditions	Provides the common modelling basis for evaluating heat exchange across the selected systems
Nanofluid-assisted enhancement	Modification of effective thermal properties through nanoparticle dispersion	Appropriate nanofluid model, concentration-dependent properties, and clearly stated phase assumptions	Applies directly to the CuO–water and TiO_2–water cases and the comparative twisted-tube nanofluid study

2.2 Governing equations and dimensionless parameters

The conservation of mass, momentum, and energy usually forms the foundation of the mathematical model used in CFD-based heat-transfer studies. The exact mathematical form of these governing equations depends on the assumptions used in each original study, such as steady or transient flow, laminar or turbulent flow, single-phase or multiphase flow, and whether heat conduction through solid components is coupled with fluid convection. This article critically reviews previous numerical investigations and does not formulate or solve a new set of governing equations. Instead, it analyzes and compares the modeling assumptions and performance measures reported in the original studies. Dimensionless parameters and engineering performance quantities are commonly used to interpret CFD results. The link between momentum and thermal diffusivity is represented by the Prandtl number, whereas the Reynolds number describes the relative impact of viscous and inertial factors. The heat-transfer coefficient and Nusselt number are commonly used to measure convective thermal performance. Hydraulic penalties may be reported through pressure drop, friction factor, or pumping-power requirements. Some studies also use a performance evaluation criterion (PEC) or another combined thermal–hydraulic index to assess whether improved heat transfer justifies the corresponding increase in flow resistance. Definitions and reference conditions for these quantities may differ among published studies. Therefore, the present review retains the definitions provided by each original source and does not treat differently defined performance indices as directly equivalent. Quantities that were not reported by an original study are identified as not reported rather than calculated or inferred.

Entropy-generation and Bejan-number analyses are included only as supplementary approaches reported in specialized heat-transfer studies, particularly for non-Newtonian microflows and nanofluid systems [12,13]. Table 2 provides the interpretative framework for the subsequent case-study comparison. However, the five selected studies do not necessarily employ the same performance parameters, baseline configurations, or operating conditions. Direct quantitative comparisons will therefore be made only when the reported definitions and conditions are sufficiently compatible; otherwise, the results will be discussed qualitatively, and any unavailable quantity will be marked as "not reported."

2.3 Turbulence and numerical modeling framework

Most turbulent heat transfer numerical models are based on the five selected case studies and do not employ an identical numerical framework. Their geometries, working fluids, operating conditions, turbulence treatments, and validation procedures differ according to the physical system being investigated. Consequently, no single turbulence model or numerical scheme can be presented as universally applicable to all five cases. The present review therefore evaluates each modeling approach according to the information reported in its original source. The principal information considered includes the CFD software or solver, dimensional representation, flow regime, turbulence model, nanofluid treatment, mesh-independence assessment, convergence criteria, and validation procedure. For conjugate heat-transfer problems, the treatment of coupling between the solid and fluid domains is also relevant because the selected coupling strategy can affect numerical stability and accuracy.

Table 2. General governing principles and performance quantities used to interpret the reviewed CFD studies

Governing principle or quantity	General role in CFD heat-transfer analysis	Treatment in the present review
Conservation of mass	Maintains flow continuity within the computational domain.	The assumptions and formulation reported by each original study are retained; no new equation is solved.
Conservation of momentum	Describes the relationship among velocity, pressure, viscous effects, and applied forces.	Source-specific flow models, turbulence treatments, and assumptions are considered where reported.
Conservation of energy	Describes heat transport within the working fluid.	Thermal properties, heat sources, and boundary conditions are recorded from the original studies where available.
Solid heat conduction and conjugate heat transfer	Represents heat conduction through solid components and its interaction with fluid convection.	Considered only for studies that explicitly model solid and fluid domains.
Reynolds number	Characterizes the relative influence of inertial and viscous effects and assists in identifying the flow regime.	Reported ranges are recorded only when provided by the original source.
Prandtl number	Relates momentum diffusivity to thermal diffusivity.	Used as supporting information where it is reported or required to interpret the working fluid.
Nusselt number or heat-transfer coefficient	Quantifies convective heat-transfer performance.	The thermal-performance measure reported by each source is extracted without redefining it.
Pressure drops, friction factor, or pumping power	Represents the hydraulic or energy penalty associated with an enhancement technique.	The source-reported measure and operating conditions are retained during comparison.
Performance evaluation criterion or combined thermal–hydraulic index	Evaluates heat-transfer improvement together with the associated hydraulic penalty.	Used only when reported by the original study, following that study's definition and reference condition.
Entropy-generation or Bejan-number measure	Evaluates thermodynamic irreversibility in specialized heat-transfer analyses [12,13].	Treated as supplementary information and not assumed to have been evaluated in all five case studies.

In nanofluid simulations, particular attention must be given to whether the nanoparticles and base fluid are represented through a single-phase homogeneous model or a multiphase formulation. Table 3 connects these numerical considerations directly with the five case studies. Only modeling details explicitly reported by the respective original sources are included. Where a particular numerical or validation detail is not reported, it is not inferred or attributed to the study. The comparison demonstrates that the credibility of a CFD result cannot be determined from the turbulence-model name alone. Mesh sensitivity, convergence behavior, boundary conditions, material-property assumptions, and validation evidence must also be considered. Accordingly, the subsequent case-study discussion distinguishes between experimentally validated results, comparisons with previously published numerical or empirical data, and results with limited validation information.

3. Advances in CFD-based heat-transfer enhancement

CFD enables the systematic evaluation of heat-exchanger geometries, operating conditions, and enhancement strategies by resolving the associated flow and temperature fields. Recent developments have extended its application from conventional performance evaluation to optimization-assisted design, in which flow paths and heat-transfer surfaces can be modified computationally [19]. Nevertheless, CFD-based design does not eliminate the need for numerical verification and experimental validation.

3.1 Topology Optimization

Topology optimization determines the arrangement of solid and fluid regions according to defined thermal and hydraulic objectives. Density-based and level-set methods can generate heat-transfer surfaces and flow passages without restricting the design to predetermined geometric configurations. For two-fluid heat exchangers, recent methods also maintain physical separation between the hot and cold streams while maximizing heat transfer under specified pressure-drop constraints [20,21]. CFD-coupled optimization has additionally been applied to heat-exchanger-fin design [22]. These approaches are relevant because they can find configurations that conventional parameter variation might miss. However, they are limited by computational complexity, dependence on objective functions and boundary conditions, and the challenge of translating complex numerical shapes into physical parts. Before practical implementation, consider minimum wall thickness, structural integrity, hydraulic penalties, and off-design performance.

3.2 Multiscale CFD

Multiscale CFD can be applied to compact heat exchangers with small features such as fins, dimples, or other small turbulators that are too fine to be resolved across the entire device. In this method, local heat-transfer and pressure-drop behavior is characterized using detailed microscale simulations. The obtained relationships are then used in reduced macroscale models for a full-device analysis.

Table 3. Numerical modeling and validation approaches reported in the selected case studies

Case study and reference	Numerical framework explicitly reported	Conditions and outputs examined	Application in the present review
Case Study I: CuO–water nanofluid in a shell-and-tube heat exchanger [14]	ANSYS Fluent; steady turbulent flow; single-phase homogeneous nanofluid representation; RANS formulation with the realizable k–ε turbulence model	CuO concentrations of 0.1–1.0 vol.% and Reynolds numbers of approximately 17,000–71,000; heat-transfer coefficient, pressure drop, and performance index	Used to evaluate the thermal improvement and hydraulic penalty associated with increasing nanoparticle concentration and Reynolds number.
Case Study II: CFD-based shell-and-tube heat-exchanger optimization [15]	Turbulent gas-phase circulation model using the standard k–ε formulation; modified exchanger configuration incorporating circulation features	Effects of inlet velocity on temperature distribution, heat-transfer behaviour, and pressure drop; simulation results supported by verification experiments	Used to examine how flow redistribution and geometric modification affect shell-and-tube heat-exchanger performance.
Case Study III: TiO_2–water nanofluid in a double-pipe heat exchanger [16]	ANSYS Fluent; two-dimensional axisymmetric, steady, incompressible, single-phase model; SST k–ω turbulence model; structured mapped mesh	Reynolds numbers of 4,000–18,000 and TiO_2 concentrations of 0.2–0.6 vol.%; heat-transfer coefficient, Nusselt number, and pressure drop	Used as a case involving documented grid-independence assessment and comparison with experimental data.
Case Study IV: Twisted-tube heat exchanger using different nanofluids [17]	Three-dimensional CFD analysis of twisted-tube configurations with different pitch lengths and nanofluids	Pitch lengths of 180, 135, 90, 67.5, and 45 mm and flow rates of 0.5–2.0 kg/s; heat-transfer performance, outlet temperature, and pressure drop	Used to assess the combined influence of geometric modification and nanofluid selection. The reported results were compared with previously published experimental and numerical evidence.
Case Study V: Ground-air heat exchanger incorporating turbulators [18]	CFD comparison of baseline PVC-pipe and turbulator-equipped configurations	Air velocities of 0.1–5.0 m/s, inlet temperatures of 0–16 °C, and a surrounding-soil temperature of 18.8 °C; heat-transfer rate	Used to examine passive flow disruption in a ground-air heat exchanger. Numerical details not explicitly reported in the source are not inferred in this review.

The procedure has been used for automotive plate heat exchangers with offset-strip fins and dimple-type turbulators [23]. The major benefit of multiscale CFD is linking local enhancement mechanisms to overall heat exchanger performance while reducing computational requirements. However, its accuracy depends on the quality and applicability of the microscale data and upscaled correlations. Models generated for one geometry, flow direction, and/or Reynolds-number range should not be used for significantly different conditions without further testing. Assessment of mesh-independence, convergence monitoring, well-defined boundary conditions, and comparison with experimental or benchmark evidence are all critical to the reliability of both topology-optimized and multiscale CFD models. Numerical and experimental investigations of shell-and-tube heat exchangers demonstrate the importance of validating predicted thermal and hydraulic behavior under application-relevant conditions [24].

4. Review methodology and case-study selection

4.1 Review design and search framework

This article employs a structured, targeted critical-review approach to compare representative CFD-based heat-transfer enhancement studies. It is not presented as an exhaustive systematic review or meta-analysis. The selection process was designed to identify well-documented studies of various heat-exchanger configurations and enhancement mechanisms for cross-system engineering comparison. During manuscript preparation, the five-study corpus was reassessed using combinations of the terms "computational fluid dynamics," "CFD," "heat-transfer enhancement," "heat exchanger," "nanofluid," "shell-and-tube," "double-pipe," "twisted tube," "turbulator," and "ground-air heat exchanger."

The five selected case studies were published between 2022 and 2025, and we included publications before and after these dates to provide the theoretical background and discuss relevant numerical and design developments. The article is not an exhaustive systematic review, and we do not provide a PRISMA-like screening count because it is a targeted critical synthesis.

4.2 Eligibility and case-study selection

Studies were acceptable for the comparative analysis if CFD was used as the primary tool to examine heat-transfer enhancement in a clearly defined engineering heat-exchanger system. Studies were also eligible if they examined a specific enhancement technique, including nanofluids, geometry modification, or passive flow disruption, and provided quantitative thermal-performance data and, if available, hydraulic-performance indicators. Sufficient information on operating conditions, numerical assumptions, and/or validation was also needed to allow for critical interpretation. Studies that were purely experimental, general review articles, or lacked extractable performance outcomes, or that were the same configuration as another study already selected as a principal case but offered no novel enhancement strategy, or that addressed a system outside the scope of the comparison were not selected as principal cases. Five studies were selected to provide variation in heat-exchanger configuration and enhancement mechanism. These comprise a CuO–water nanofluid in a shell-and-tube heat exchanger [14], CFD-based optimization of a shell-and-tube heat exchanger [15], a TiO_2–water nanofluid in a double-pipe heat exchanger [16], twisted-tube configurations using different nanofluids [17], and a ground-air heat exchanger incorporating internal turbulators [18]. The cases are treated as representative examples for critical comparison and are not claimed to constitute the complete CFD heat-transfer enhancement literature.

4.3 Data extraction and comparative synthesis

Information was extracted from each selected source using a common comparison framework. The extracted information included the heat-exchanger configuration, working fluid, enhancement strategy, reported CFD software or numerical approach, flow regime, operating range, thermal-performance indicator, pressure-drop or other hydraulic measure, validation method, and principal source-reported limitations. Results were kept as defined by the reference conditions used in each original study. If the required raw data or appropriate baseline conditions were not available, then the values were not recalculated or normalized. If a source did not report a parameter, then it was recorded as “not reported” and not estimated. Only quantitative comparisons were made if the definitions and operating conditions were sufficiently compatible; otherwise, a synthesis of the evidence was performed qualitatively. The comparative assessment was based on the following four categories: thermal improvement, hydraulic penalty, numerical validation, and practical applicability. These were used to determine the relative strengths, weaknesses, and constraints of the enhancement strategies for the five engineering systems chosen.

5. Case study I: shell-and-tube heat exchanger using CuO–water nanofluid

Cruz et al. [14] numerically investigated CuO–water nanofluid flowing through the shell side of a shell-and-tube heat exchanger. Their steady-state ANSYS Fluent model employed a single-phase homogeneous representation and the realizable k–ε turbulence model. CuO nanoparticles with an average diameter of 29 nm were investigated at Reynolds numbers of roughly 17,800–71,100 and concentrations between 0.1 and 1.0 vol.%. The heat-transfer coefficient, pressure drop, temperature distribution, and integrated thermal-hydraulic performance were all assessed.

The heat-transfer coefficient increased by roughly 9–12% at a CuO concentration of 0.1 vol.%, while it improved by roughly 41-48% at a concentration of 1.0 vol.%, according to the figures published by the source. In contrast, the pressure drop at 1.0 vol.% was roughly twice that of water. For particle concentrations below 0.25 vol.%, performance-index values greater than unity were recorded across the investigated Reynolds numbers, suggesting that lower particle concentrations offered a more advantageous balance between hydraulic penalty and thermal benefit. Figure 1 displays the numerical connections between heat-transfer enhancement, pressure drop, Reynolds number, and nanoparticle loading as provided by the source. Figure 1 shows that increasing CuO nanoparticle loading consistently enhanced the heat-transfer coefficient but also increased the pressure-drop penalty. It also shows that the percentage enhancement was greater at lower Reynolds numbers, whereas the absolute pressure drop increased markedly with Reynolds number. For validation, the water-based simulation results were compared with an earlier CFD model that had been experimentally and theoretically validated. Nevertheless, the pressure drop was underpredicted by approximately 20%. Furthermore, the single-phase model excluded particle aggregation, sedimentation, fouling, and other long-term nanofluid effects. These limitations should be considered when interpreting the reported performance for practical heat-exchanger operation [14].

6. Case study II: CFD-based optimization of shell-and-tube heat exchanger

Wang et al. [15] numerically and experimentally examined heat transfer in a gas-phase circulatory shell-and-tube heat exchanger. To enhance shell-side gas circulation and expand the accessible heat-transfer surface, their ANSYS-based numerical model included 48 triangular heat-transfer tubes and a spiral circulation blade. The conventional k–ε turbulence model was used to solve the mass, momentum, and energy conservation equations.

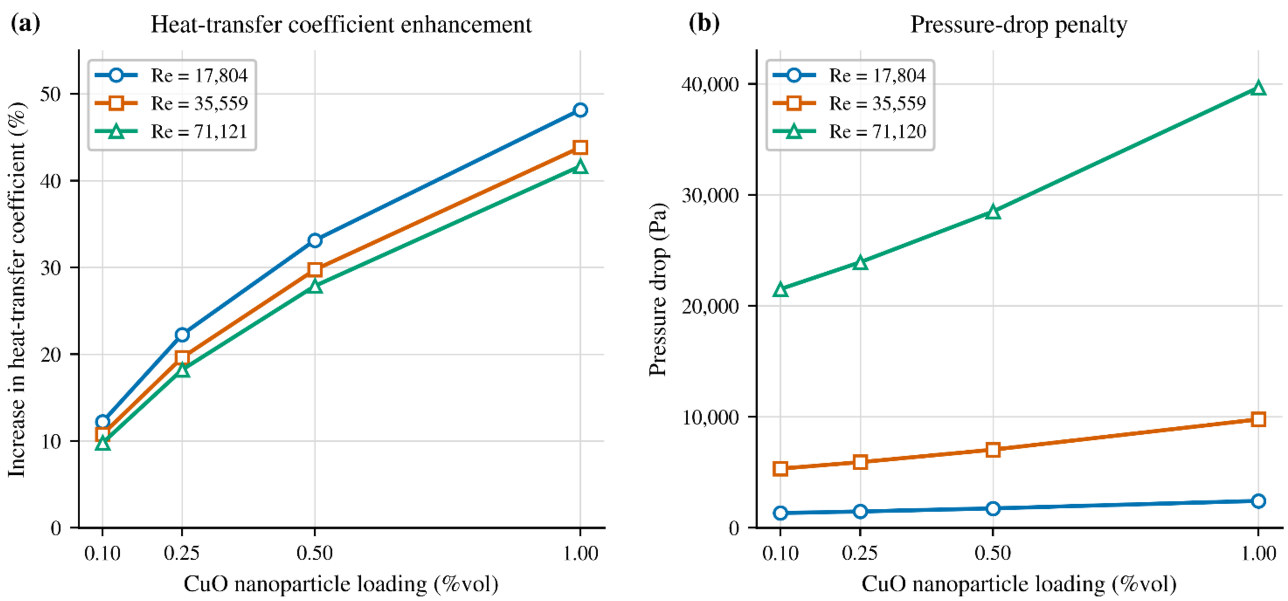

Figure 1. Effect of CuO nanoparticle loading and Reynolds number on (a) the percentage increase in the heat-transfer coefficient and (b) the pressure drop in a shell-and-tube heat exchanger

The mesh-independence assessment indicated that the predicted temperature became stable at a mesh count of approximately 5.2 million, with the reported temperature variation remaining within 4°C. The simulation was verified experimentally at hot-air inlet velocities of 3, 6, 9, and 12 m/s, with ten measurements averaged for each operating condition. Figure 2 compares the source-reported experimental and simulated pressure-drop values. The experimental and numerical pressure drops were 756, 763, 819, and 810 Pa at inlet velocities of 3, 6, 9, and 12 m/s, respectively. The maximum experimental pressure drop was 819 Pa at 9 m/s, and it was slightly lower at 12 m/s. The study also found that the minimum heat-transfer error is 5.1% at 6 m/s, and the maximum error is 9.87% at 9 m/s.

The Reynolds number analysis was carried out for numbers between 35,000 and 95,000. The pressure drops on the shell side and the heat-transfer coefficient generally increased with Reynolds number, with the rate of increase in pressure drop becoming smaller at about Re = 85,000. Tube size and spiral-blade spacing were also important design parameters. However, the authors investigated only four inlet-velocity conditions and reported no performance evaluation criterion, pumping-power comparison, economic assessment, or formal multi-objective optimization. Hence, the study can serve as a parameter-screening tool using CFD methods, but it does not support the broad claim that exchanger efficiency was enhanced without a significant penalty in hydraulic performance. Figure 2 shows that the experimental pressure-drop values and the CFD results are close, and that the hydraulic penalty did not increase monotonically but instead peaked at an intermediate velocity.

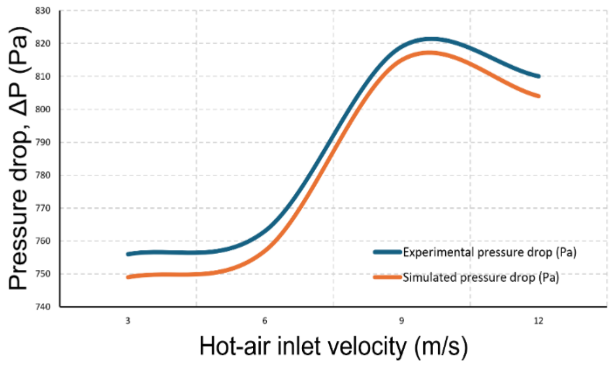

Figure 2. Experimental and simulated pressure drop across the gas-phase circulatory shell-and-tube heat exchanger at the hot-air inlet

7. Case study III: double pipe heat exchanger with TiO_2–water nanofluid

Alhulaifi [16] numerically evaluated turbulent forced-convection heat transfer in a horizontal counterflow double-pipe heat exchanger using TiO_2–water nanofluids. The two-dimensional axisymmetric model was developed using ANSYS Fluent 15.0 and employed a steady, incompressible, single-phase formulation with the shear-stress-transport k–ω turbulence model. The heat exchanger comprised an inner copper tube, an outer PVC pipe, and a total flow length of 1.5 m. A structured mapped grid containing 105,840 elements and 106,555 nodes was selected following a grid-independence assessment, and convergence was evaluated using residual criteria of 10^{-6} for the governing equations.

TiO_2 nanoparticles with a diameter of 21 nm were examined at concentrations of 0.2, 0.4, and 0.6 vol.% over Reynolds numbers ranging from 4,000 to 18,000. The nanofluid entered the inner tube at 298.15 K, while hot water flowed through the annular region at 3 L/min and temperatures between 308.15 and 323.15 K. The numerical Nusselt number and pressure-drop predictions were compared with previously published experimental data. The comparison indicated satisfactory agreement, particularly for the 0.2 vol.% nanofluid condition, while pressure drop increased with Reynolds number.

Increasing Reynolds number and nanoparticle concentration increased the convective heat-transfer coefficient. At Re = 16,200, the reported improvements relative to water were 1.9%, 2.7%, and 3.5% for TiO_2 concentrations of 0.2, 0.4, and 0.6 vol.%, respectively. Figure 3 presents these concentration-dependent improvements. Increasing the annular hot-water temperature increased the heat-transfer rate but did not substantially change the nanofluid heat-transfer coefficient. Although the results demonstrate a consistent thermal benefit, the reported improvements were comparatively modest. The study did not provide a complete performance evaluation criterion or pumping-power comparison across all nanoparticle concentrations. Moreover, the single-phase model did not resolve particle slip, agglomeration, sedimentation, stability, or long-term fouling. Consequently, the reported thermal improvements should not be interpreted as evidence of an equivalent overall thermal–hydraulic or operational advantage without further experimental evaluation. Figure 3 demonstrates that increasing the TiO_2 concentration from 0.2 to 0.6 vol.% progressively increased the convective heat-transfer coefficient enhancement from 1.9% to 3.5%; however, these modest gains must be considered alongside the corresponding increase in viscosity and the absence of a complete pumping-power assessment.

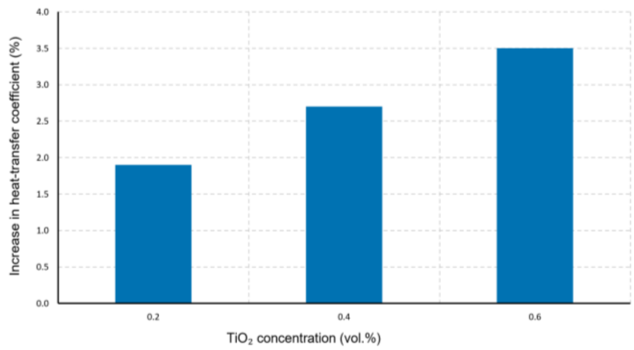

Figure 3. Source-reported increase in convective heat-transfer coefficient relative to water for TiO_2–water nanofluids at Re = 16,200

8. Case study IV: twisted tube heat exchanger with nanofluids

Ghazanfari et al. [17] employed three-dimensional CFD to examine the combined effects of twisted-tube geometry and water-based Al_2O_3, Cu, CuO, and TiO_2 nanofluids on heat-exchanger performance. Five twist-pitch lengths—180, 135, 90, 67.5, and 45 mm- were investigated over mass flow rates ranging from 0.5 to 2.0 kg/s. The numerical method was validated through comparison with experimental and numerical data reported in previous studies. The results indicated that twisted tubes and nanofluids enhanced heat transfer, although nanofluids produced a slight increase in pressure drop. The 45 mm twist pitch was identified as the optimized configuration within the investigated range.

Compared with the smooth-tube heat exchanger fitted with six baffles, the twisted tube with a pitch of 45 mm produced source-reported heat-transfer improvement factors of 1.04 using 0.1 vol.% Cu nanofluid and 1.12 using 0.15 vol.% Al_2O_3 nanofluid. Replacing the baffles with the optimized twisted-tube configuration also reduced the pressure drop by approximately 1.55. Figure 4 presents the source-reported heat-transfer improvement factors for the optimized twisted-tube configuration. Figure 4 demonstrates that the Al_2O_3–water nanofluid at 0.15 vol.% provided the greater heat-transfer improvement under the reported conditions, although the two results were obtained at different nanoparticle concentrations and should therefore not be interpreted as a concentration-controlled comparison.

9. Case study V: ground-air heat exchanger with internal turbulators

Jalili and Jalili [18] employed CFD to evaluate heat-transfer enhancement produced by internal turbulators in a PVC-pipe ground-air heat exchanger intended for heating applications. The numerical investigation considered airflow velocities ranging from 0.1 to 5 m/s and inlet-air temperatures from 0 to 16 °C. The surrounding soil temperature was maintained at 18.8 °C.

Simulations of the smooth pipe were first performed to establish the baseline heat-transfer rate, after which the turbulator-equipped configuration was evaluated under corresponding operating conditions. The source-reported results demonstrated that the relative influence of the turbulators was greater at the lower of the two quantified velocities. At an airflow velocity of 2 m/s, the heat-transfer rate increased from 841.386 W in the smooth pipe to 987.128 W with turbulators, representing an improvement of 17.31%. At 5 m/s, the corresponding heat-transfer rate increased from 2253.135 to 2596.305 W, equivalent to an improvement of 15.22%. Although the absolute heat-transfer rate was higher at 5 m/s, the turbulators produced a greater percentage improvement at 2 m/s. Figure 5 compares the source-reported heat-transfer rates for the smooth and turbulator-equipped configurations. Figure 5 shows that incorporating turbulators increased the heat-transfer rate at both airflow velocities, with a comparatively greater percentage improvement at the lower velocity. These findings indicate that internal turbulators may be particularly beneficial for low-velocity ground-air heating applications. However, the reported thermal improvement should not automatically be interpreted as a complete thermal–hydraulic advantage because the additional flow resistance and fan-power requirement associated with the turbulators must also be considered.

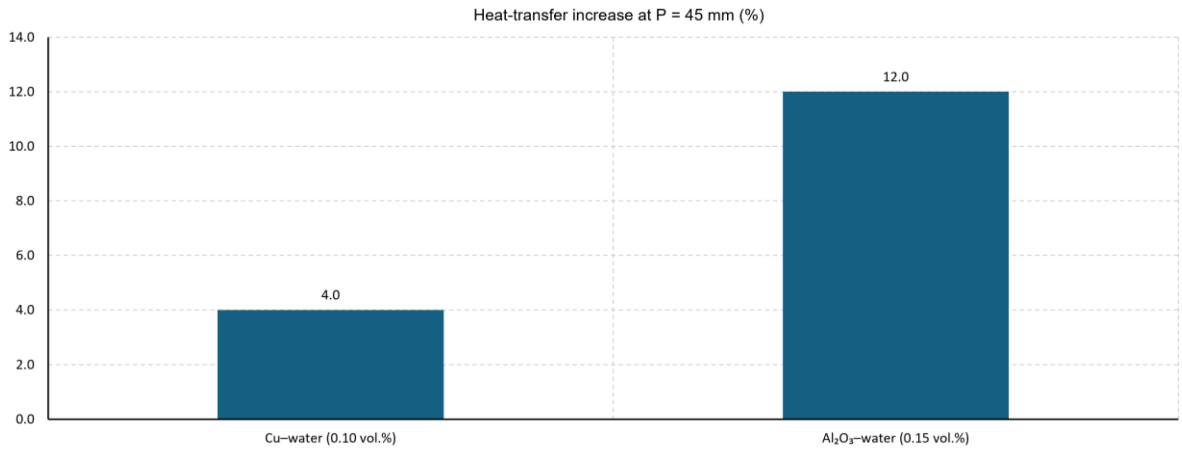

Figure 4. Heat-transfer improvement factors obtained using Cu–water and Al_2O_3–water nanofluids in a twisted tube

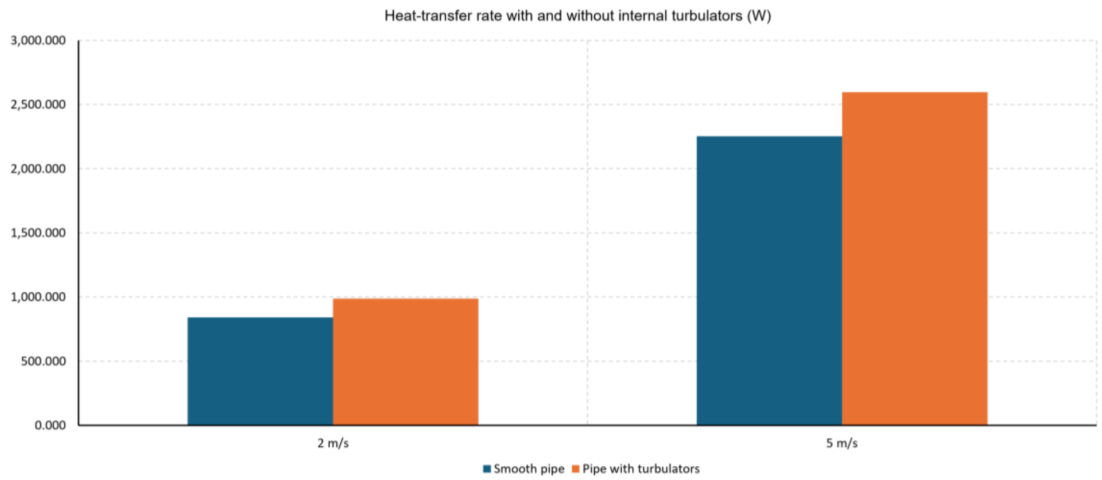

Figure 5. Heat-transfer rates of smooth and turbulator-equipped PVC pipes in a ground-air

Furthermore, the constant soil-temperature assumption does not fully represent seasonal variations in soil temperature, moisture content, and long-term thermal saturation. Consequently, experimental validation under variable field conditions is required before recommending the configuration for large-scale implementation.

10. Comparative analysis across case studies

The five case studies represent different heat-exchanger configurations, enhancement mechanisms, working fluids, operating ranges, baseline conditions, and performance indicators. Their numerical results therefore cannot be interpreted as directly equivalent or used to establish a universal ranking. Nevertheless, a structured comparison can identify the relative thermal benefits, hydraulic penalties, validation strengths, and practical limitations associated with each enhancement strategy. Table 4 presents the principal quantitative and methodological information extracted from the selected studies. Values that were unavailable in the original sources are identified as "Not reported" rather than estimated.

10.1 Cross-case thermal–hydraulic comparison

The comparison demonstrates that the magnitude of thermal enhancement depends strongly on the enhancement mechanism and reference condition. The CuO–water shell-and-tube case produced the largest reported relative increase among the extracted nanofluid results, reaching approximately 41–48% at 1.0 vol.% CuO [14]. However, this improvement was accompanied by a pressure drop approximately twice that of water. Performance-index values greater than unity were mainly associated with concentrations below 0.25 vol.%, indicating that maximizing nanoparticle concentration does not necessarily provide the most favorable thermal–hydraulic performance. The gas-phase circulatory shell-and-tube study provided the most direct experimental comparison of pressure-drop predictions among the five cases [15]. Experimental and numerical pressure-drop values remained close at the four tested inlet velocities. Nevertheless, the study did not report a common percentage improvement in heat transfer, PEC, pumping-power comparison, or economic assessment. It consequently provides useful evidence of numerical agreement and parameter sensitivity but does not establish formal multi-objective optimality.

The TiO_2–water double-pipe case reported comparatively modest improvements of 1.9–3.5% at Re = 16,200 [16]. These gains increased with nanoparticle concentration, but the study did not provide a complete pumping-power or PEC assessment. Consequently, the CuO and TiO_2 results cannot be compared solely according to their reported heat-transfer percentages. The exchanger geometries, nanoparticle materials, concentrations, Reynolds-number ranges and reference conditions were different, and both studies used single-phase formulations that excluded several long-term particle effects. The twisted-tube case demonstrates the potential benefit of combining geometric modification with nanofluids [17]. At a twist pitch of 45 mm, the source-reported improvement factors corresponded to approximately 4% for 0.1 vol.% Cu–water and 12% for 0.15 vol.% Al_2O_3–water. The optimized twisted-tube configuration also produced a reported pressure-drop reduction relative to the baffled baseline. Nevertheless, the two nanofluids were evaluated at different concentrations, so the result should not be interpreted as a concentration-controlled comparison of nanoparticle materials.

Internal turbulators increased the heat-transfer rate of the ground-air heat exchanger by 17.31% at 2 m/s and 15.22% at 5 m/s [18]. The larger percentage improvement at the lower velocity suggests that passive flow disruption may be particularly beneficial in low-velocity heating conditions. However, pressure drop and fan power were not quantified. The reported thermal improvement therefore cannot establish an overall energy-efficiency advantage without an accompanying hydraulic assessment.

10.2 Numerical validation and modeling reliability

The CuO–water study compared its water-based results with an earlier experimentally and theoretically validated CFD configuration [14]. However, pressure drop was underpredicted by approximately 20%, which is important when assessing the reported thermal–hydraulic performance. The validation evidence differed substantially across the selected studies. The gas-phase shell-and-tube model was directly compared with experiments at inlet velocities of 3, 6, 9 and 12 m/s [15]. The reported numerical pressure drops differed only slightly from the experimental values, although heat-transfer errors ranged from 5.1% to 9.87%. The TiO_2–water study documented grid independence and compared its numerical predictions with previously published experimental results [16]. These procedures increase confidence in the reported trends, but they do not eliminate uncertainty associated with the single-phase nanofluid formulation. The twisted-tube model was evaluated against experimental and numerical evidence from previous studies rather than through a dedicated experiment under every investigated nanofluid and pitch condition [17]. For the ground-air heat exchanger, the extracted evidence did not provide sufficient detail to establish equivalent experimental validation under variable field conditions [18].

The nanofluid cases also demonstrate a common modeling limitation. Single-phase homogeneous formulations assume thermal and velocity equilibrium between the nanoparticles and base fluid. Although this assumption reduces computational cost, it does not explicitly represent particle migration, slip, aggregation, sedimentation, or concentration redistribution. Predictions obtained using such models should therefore be interpreted as idealized thermal–hydraulic estimates rather than complete representations of long-term nanofluid operation. Only the CuO–water study reported a combined performance index, and even that indicator was specific to its own reference conditions [14]. The remaining studies generally reported heat-transfer and pressure-drop behavior separately or omitted a normalized combined criterion. This inconsistency prevents direct ranking across systems and demonstrates the need for standardized reporting of heat-transfer improvement, pressure drop, pumping power, PEC, uncertainty, and validation error.

10.3 Emerging design directions: topology optimization and multiscale CFD

Topology optimization extends CFD-based heat-exchanger design beyond conventional parameter variation by allowing the computational arrangement of fluid passages and solid conduction regions. Two-fluid topology-optimization methods can maintain physical separation between hot and cold streams while seeking configurations that reduce thermal resistance under hydraulic constraints [20,21]. Such methods may identify non-intuitive channel arrangements, but their results are sensitive to boundary conditions, objective functions, mesh resolution, and the constraints imposed during optimization.

Table 4. Quantitative and methodological comparison of the five selected CFD case studies

Case study and reference	Enhancement strategy and operating range	Reported thermal improvement	Hydraulic penalty and PEC	Numerical model	Validation approach	Principal limitation
CuO–water nanofluid in a shell-and-tube heat exchanger [14]	CuO concentrations of 0.1–1.0 vol.% at Re ≈ 17,800–71,100	Approximately 9–12% at 0.1 vol.% and 41–48% at 1.0 vol.%	At 1.0 vol.%, pressure drop was approximately twice that obtained using water. Performance-index values greater than unity were reported primarily below 0.25 vol.%	Steady ANSYS Fluent model; single-phase homogeneous representation; realizable k–ε turbulence model	Water-based results were compared with a previously validated CFD configuration; pressure drop was underpredicted by approximately 20%	Single-phase model excluded particle aggregation, sedimentation, fouling and long-term stability
Gas-phase circulatory shell-and-tube heat exchanger [15]	Spiral circulation blade and 48 triangular tubes; inlet velocities of 3, 6, 9 and 12 m/s; Re ≈ 35,000–95,000	Heat-transfer coefficient increased with Reynolds number, but a common percentage enhancement was not reported	Experimental pressure drop ranged from 756 to 819 Pa, while the corresponding numerical values ranged from 749 to 815 Pa. PEC and pumping-power comparison were not reported	ANSYS-based model; standard k–ε turbulence model; mesh count of approximately 5.2 million	Direct comparison with experiments; reported heat-transfer error ranged from 5.1% to 9.87%	Only four inlet velocities were experimentally examined; no multi-objective, economic or pumping-power assessment
TiO_2–water nanofluid in a double-pipe heat exchanger [16]	TiO_2 concentrations of 0.2–0.6 vol.% at Re = 4,000–18,000	At Re = 16,200, improvements of 1.9%, 2.7% and 3.5% were reported for 0.2, 0.4 and 0.6 vol.%, respectively	Pressure drop increased with Reynolds number. A normalized pressure-drop penalty, PEC and complete pumping-power comparison were not reported	ANSYS Fluent 15.0; two-dimensional axisymmetric single-phase model; SST k–ω turbulence model; 105,840 elements	Grid-independence assessment and comparison with previously published experimental data	Modest thermal improvement; particle slip, agglomeration, sedimentation, fouling and long-term stability were not represented
Twisted-tube heat exchanger with nanofluids [17]	Twist pitches of 180–45 mm and mass-flow rates of 0.5–2.0 kg/s; optimized pitch of 45 mm	Approximately 4% using 0.1 vol.% Cu–water and 12% using 0.15 vol.% Al_2O_3–water, relative to the stated smooth-tube baseline	Nanofluids produced a slight pressure-drop increase; replacing the baffled baseline with the optimized twisted-tube configuration produced a source-reported pressure-drop reduction factor of approximately 1.55. A common PEC was not reported	Three-dimensional CFD analysis of twisted-tube configurations	Comparison with experimental and numerical evidence reported in previous studies	Nanofluids were compared at different concentrations; manufacturability, fouling and long-term operation were not assessed
Ground-air heat exchanger with internal turbulators [18]	Air velocities of 0.1–5.0 m/s; inlet temperature of 0–16 °C; soil temperature maintained at 18.8 °C	Heat-transfer rate increased by 17.31% at 2 m/s and 15.22% at 5 m/s	Pressure-drop and fan-power penalties were not quantified; PEC was not reported	CFD comparison of smooth and turbulator-equipped PVC pipes	Detailed experimental-validation information was not reported in the extracted evidence	Constant soil-temperature assumption; seasonal soil variation, fan power and long-term field performance were not evaluated

Algorithmic optimization has also been applied to heat-exchanger fins used in aerospace systems, where heat-transfer performance must be balanced against mass and geometric restrictions [22]. However, a numerically optimal configuration may not be practically optimal when minimum wall thickness, structural integrity, manufacturability, fouling tolerance, maintenance access, and off-design performance are considered.

These constraints should therefore be incorporated into the optimization formulation rather than evaluated only after a design has been generated. Multiscale CFD provides another route for analyzing compact heat exchangers containing fins, dimples, or small turbulators that would be computationally expensive to resolve throughout an entire device. Detailed local simulations can derive microscale heat-transfer and pressure-drop information, which is then incorporated into a reduced macroscale model. This approach has been applied to automotive plate heat exchangers containing offset-strip fins and dimple-type turbulators [23]. Its reliability, however, depends on the validity of the microscale information and the range over which the upscaled relationships are applied. Experimental comparison remains essential for both topology-optimized and multiscale designs. Numerical and experimental analysis of shell-and-tube exchangers with different tube geometries demonstrates that geometric changes can affect thermal and hydraulic behavior simultaneously [24]. Future optimization frameworks should therefore combine thermal performance, pressure loss, structural and manufacturing constraints, uncertainty analysis, and application-relevant experimental validation.

11. Engineering implications and design guidelines

The comparative evidence indicates that selection of a heat-transfer enhancement strategy should be based on the required thermal performance, acceptable pressure loss, available pumping or fan power, geometric constraints, manufacturing capability, maintenance requirements, and expected operating life. Heat-transfer improvement alone is insufficient for design selection because an enhancement that provides a high thermal gain may become inefficient when its hydraulic and operational penalties are considered. For nanofluid-assisted systems, the CuO–water case indicates that lower nanoparticle concentrations may provide a more favorable balance between heat-transfer improvement and pressure-drop penalty than the maximum concentration. Concentrations below 0.25 vol.% produced performance-index values greater than unity under the examined conditions, whereas the 1.0 vol.% concentration substantially increased pressure drop. The TiO_2–water case produced smaller thermal gains and did not include a complete pumping-power assessment. Nanofluids should therefore be considered where geometric modification is limited and moderate thermal enhancement is useful, but only after stability, viscosity, sedimentation, fouling, material compatibility, and pumping requirements have been evaluated experimentally.

Geometric modification may be more suitable when the exchanger structure can be redesigned and manufactured without unacceptable cost or maintenance difficulty. The twisted-tube configuration demonstrated that secondary-flow generation can improve heat transfer while reducing pressure drop relative to the stated baffled baseline. However, practical adoption requires confirmation of fabrication tolerances, structural durability, cleaning access, fouling behavior, and performance away from the optimized operating condition. Similarly, the gas-phase circulatory shell-and-tube configuration showed close agreement between experimental and numerical pressure-drop values, but its design should not be described as globally optimal without a combined assessment of heat transfer, pumping power, size, cost, and operating range. Internal turbulators may provide useful enhancement in low-velocity ground-air heating applications because their relative thermal effect was greater at 2 m/s than at 5 m/s. Nevertheless, the absence of quantified pressure-drop and fan-power data prevents a complete energy assessment. Turbulator geometry, spacing and blockage ratio should therefore be selected through coupled thermal–hydraulic analysis and subsequently validated under variable soil temperature, moisture and seasonal operating conditions.

CFD models intended to support engineering decisions should include a documented mesh-independence assessment, convergence criteria, material-property assumptions, turbulence or multiphase model justification and comparison with experimental or benchmark data. Validation should cover both thermal and hydraulic quantities because agreement in outlet temperature does not necessarily ensure accurate pressure-drop prediction. When using a PEC or another combined metric, explicitly report its equation, reference configuration, and operating conditions. From a lifecycle perspective, evaluate an enhancement strategy in relation to energy consumption, material use, fabrication complexity, maintenance frequency, reliability, and economic feasibility. Passive enhancement devices do not require a separate energy source, but they can increase the pumping or fan power needed to maintain flow. Sustainable heat-exchanger design therefore requires optimizing total system performance rather than maximizing a single heat-transfer indicator.

12. Limitations and future research directions

The first limitation concerns the comparability of the selected evidence. The five studies examined different exchanger configurations, working fluids, operating ranges, enhancement mechanisms, baseline conditions, and performance measures. Reported percentages and performance factors therefore cannot be combined as though they were obtained under equivalent conditions. The absence of a common PEC, pumping-power measure, and uncertainty-reporting framework also prevents a formal quantitative ranking of the five enhancement strategies. The present article should consequently be interpreted as a targeted critical comparison rather than a meta-analysis.

The nanofluid studies were primarily based on single-phase homogeneous models [14,16]. These models assume thermal and velocity equilibrium between nanoparticles and the base fluid and do not explicitly represent particle migration, slip, agglomeration, sedimentation, or local concentration changes. Long-term nanofluid stability, fouling, erosion, corrosion, material compatibility, and changes in thermophysical properties were not comprehensively assessed. Consequently, short-duration CFD predictions cannot independently demonstrate reliable long-term performance in an operational heat exchanger. Hydraulic and energy penalties were also reported inconsistently. Some studies described pressure-drop trends without providing a normalized penalty, pumping-power comparison, or combined performance criterion. The ground-air heat-exchanger study quantified the increase in heat-transfer rate but did not provide the corresponding pressure-drop or fan-power requirement [18]. Without these quantities, it is not possible to determine whether the additional thermal output compensates for the energy required to maintain airflow. Geometric enhancement adds other practical constraints. Twisted tubes, internal turbulators, spiral circulation features, and topology-generated flow paths can enhance mixing and disrupt the boundary layer, but these features may also increase manufacturing complexity, inspection complexity, and maintenance requirements. Fouling may be more likely in complex internal geometries, and it may be hard to clean or repair. These must therefore be taken into account in the numerical optimization, along with the minimum feature size, structural integrity, method of fabrication, surface quality, maintenance access, and operation off design.

A considerable drawback is still the lack of validation. The selected studies used a variety of mesh-independence assessments, comparisons with published correlations, and benchmarking and direct experimentation. Validation was not necessarily performed for each operating condition, nanofluid concentration, and geometric configuration. Future studies should provide mesh sensitivity, numerical uncertainty, and convergence behavior, as well as thermal and hydraulic validation and experimental error. Field-scale testing is especially important for ground-air systems, as the constant soil temperature does not account for seasonal changes in soil temperature, soil moisture, and long-term thermal saturation.

In the future, nanofluid studies should be correlated with experimental studies on the stability, redistribution of concentration, fouling, erosion, and degradation of nanofluid properties during long-term operation. Decisions on using multiphase formulations should be based on comparing prediction accuracy with single-phase models and assessing whether the extra expense of multiphase models is warranted. When reporting thermal enhancement, also report pressure drop, pumping power, uncertainty, and long-term stability. In future work, geometric and topology optimization studies should consider not only the objective function but also manufacturing and operational constraints. Thermal resistance and pressure loss should be considered along with structural integrity, minimum wall thickness, materials used, fouling resistance, access for maintenance, and life cost. It is also important to test optimized configurations to off-design conditions, in addition to the operating point used in the optimization. A standardized thermal–hydraulic reporting framework is required to improve comparability across CFD heat-transfer studies. At a minimum, future investigations should report the reference configuration, Reynolds-number range, heat-transfer improvement, pressure-drop penalty, pumping or fan power, PEC definition, numerical uncertainty, validation error, and relevant economic information. This reporting would allow CFD-based enhancement strategies to be assessed by total engineering performance rather than isolated thermal improvement.

13. Conclusion

This critical review compared five published CFD case studies and found that no enhancement strategy is universally superior. Thermal gains depend on exchanger configuration, working fluid, operating conditions, and the balance between heat transfer and hydraulic penalties. Accordingly, heat-transfer improvement should not be assessed without pressure drop, pumping or fan power, validation, manufacturability, and long-term operation. The CuO–water case produced improvements of approximately 9–48%, but pressure drop at 1.0 vol.% was about twice that of water; lower concentrations provided the more favorable combined performance. The TiO_2–water case produced more modest gains of 1.9–3.5%. The optimized twisted-tube configuration provided approximately 4% and 12% enhancement for the reported Cu–water and Al_2O_3–water conditions, whereas internal turbulators increased ground-air heat transfer by 15.22–17.31%. However, missing pumping- or fan-power data limited complete energy evaluation. Low-concentration nanofluids could be considered for practical design if structural modification is not allowed and the stability, fouling, erosion, and pumping requirements are checked. Twisted tubes and turbulators might be best where fabrication, maintenance, and hydraulic penalties are acceptable. The CFD predictions need to be assessed for mesh independence and validated thermally and hydraulically. In the future, focus should be on standardizing reporting of heat-transfer enhancement, pressure drop, pumping power, PEC, uncertainty, and lifecycle cost. Multiphase nanofluid models should consider particle migration, aggregation, sedimentation, and degradation, and these effects should be validated. Topology optimization and multiscale CFD should incorporate manufacturing, structural, economic, and off-design constraints and undergo application-relevant experimental validation.

Ethical issue

The authors are aware of and comply with best practices in publication ethics, specifically regarding authorship (avoidance of guest authorship), dual submission, manipulation of figures, competing interests, and compliance with research ethics policies. The authors adhere to publication requirements that the submitted work is original and has not been published elsewhere. All survey participants provided informed consent before participating, and the anonymity and confidentiality of all respondents were strictly maintained throughout the research process.

Data availability statement

The manuscript contains all the data. However, additional data will be provided by the corresponding author upon reasonable request.

Conflict of interest

The authors declare no potential conflict of interest.

References

[1] L. Pandey and S. Singh, "Numerical Analysis for Heat Transfer Augmentation in a Circular Tube Heat Exchanger Using a Triangular Perforated Y-Shaped Insert," Fluids, vol. 6, no. 7, p. 247, Jul. 2021, doi: 10.3390/fluids6070247.

[2] D. Lahaye, P. Nakate, K. Vuik, F. Juretić, and M. Talice, "Modeling Conjugate Heat Transfer in an Anode Baking Furnace Using OpenFoam," Fluids, vol. 7, no. 4, p. 124, Apr. 2022, doi: 10.3390/fluids7040124.

[3] C. Sinn, F. Kranz, J. Wentrup, J. Thöming, G. D. Wehinger, and G. R. Pesch, "CFD Simulations of Radiative Heat Transport in Open-Cell Foam Catalytic Reactors," Catalysts, vol. 10, no. 6, p. 716, Jun. 2020, doi: 10.3390/catal10060716.

[4] X. F. Peng and G. P. Peterson, "Forced convection heat transfer of single-phase binary mixtures through microchannels," Experimental Thermal and Fluid Science, vol. 12, no. 1, pp. 98–104, Jan. 1996, doi: 10.1016/0894-1777(95)00079-8.

[5] J. Wang, S. Qi, and Y. Xu, "Numerical Investigations of the Thermal-Hydraulic Characteristics of Microchannel Heat Sinks Inspired by Leaf Veins," Energies, vol. 17, no. 2, p. 311, Jan. 2024, doi: 10.3390/en17020311.

[6] S. Mi, M. Chen, T. Li, and L. Yang, "CFD Simulation of Flow and Heat Transfer of V-Shaped Wavy Microchannels," Processes, vol. 13, no. 9, p. 2865, Sep. 2025, doi: 10.3390/pr13092865.

[7] P. Meliga, W. Abdel Nour, D. Laboureur, D. Serret, and E. Hachem, "Multi-Objective Topology Optimization of Conjugate Heat Transfer Using Level Sets and Anisotropic Mesh Adaptation," Fluids, vol. 9, no. 5, p. 105, May 2024, doi: 10.3390/fluids9050105.

[8] W. Lian, Y. Jiang, H. Chen, Y. Li, and X. Liu, "Heat Transfer Characteristics of an Aeroengine Turbine

Casing Based on CFD and the Surrogate Model," Energies, vol. 15, no. 18, p. 6743, Jan. 2022, doi: 10.3390/en15186743.

[9] O. M. Sarikaya, M. Kuzay, S. Yilmaz, and E. Demirel, "Development and Optimization of a Micro-Baffle for the Enhancement of Heat Transfer in Film Boiling," Energies, vol. 17, no. 20, p. 5224, Jan. 2024, doi: 10.3390/en17205224.

[10] P. Renze and K. Akermann, "Simulation of Conjugate Heat Transfer in Thermal Processes with Open Source CFD," ChemEngineering, vol. 3, no. 2, p. 59, Jun. 2019, doi: 10.3390/chemengineering3020059.

[11] E. Tandis, P. Cardiff, and A. Ashrafizadeh, "Analysis of Coupling Strategies for Conjugate Heat Transfer Problems," OpenFOAM® Journal, vol. 5, pp. 38–58, Mar. 2025, doi: 10.51560/ofj.v5.92.

[12] V. Anand and I. C. Christov, "On the Enhancement of Heat Transfer and Reduction of Entropy Generation by Asymmetric Slip in Pressure-Driven Non-Newtonian Microflows," J. Heat Transfer, vol. 141, no. 022403, Dec. 2018, doi: 10.1115/1.4042157.

[13] G. Liang and I. Mudawar, "Review of single-phase and two-phase nanofluid heat transfer in macro-channels and micro-channels," International Journal of Heat and Mass Transfer, vol. 136, pp. 324–354, Jun. 2019, doi: 10.1016/j.ijheatmasstransfer.2019.02.086.

[14] P. A. D. Cruz, E.-J. E. Yamat, J. P. E. Nuqui, and A. N. Soriano, "Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger," Digital Chemical Engineering, vol. 3, p. 100014, Jun. 2022, doi: 10.1016/j.dche.2022.100014.

[15] J. Wang, J. Nan, and Y. Wang, "CFD-Based Optimization of a Shell-and-Tube Heat Exchanger," Fluid Dynamics and Materials Processing, vol. 19, no. 11, pp. 2761–2775, Sep. 2023, doi: 10.32604/fdmp.2023.021175.

[16] A. S. Alhulaifi, "Computational Fluid Dynamics Heat Transfer Analysis of Double Pipe Heat Exchanger and Flow Characteristics Using Nanofluid TiO2 with Water," Designs, vol. 8, no. 3, p. 39, Jun. 2024, doi: 10.3390/designs8030039.

[17] V. Ghazanfari, A. Taheri, Y. Amini, and F. Mansourzade, "Enhancing heat transfer in a heat exchanger: CFD study of twisted tube and nanofluid (Al2O3, Cu, CuO, and TiO2) effects," Case Studies in Thermal Engineering, vol. 53, p. 103864, Jan. 2024, doi: 10.1016/j.csite.2023.103864.

[18] P. Jalili and B. Jalili, "Computational fluid dynamics simulation of enhanced heat transfer in ground-air heat exchangers using turbulators in PVC pipe systems," Case Studies in Thermal Engineering, vol. 68, p. 105949, Apr. 2025, doi: 10.1016/j.csite.2025.105949.

[19] M. A. S. de Troya, D. A. Tortorelli, J. Andrej, and V. A. Beck, "Three-dimensional topology optimization of heat exchangers with the level-set method," Nov. 18, 2021, arXiv: arXiv:2111.09471. doi: 10.48550/arXiv.2111.09471.

[20] H. Kobayashi, K. Yaji, S. Yamasaki, and K. Fujita, "Topology design of two-fluid heat exchange," Struct Multidisc Optim, vol. 63, no. 2, pp. 821–834, Feb. 2021, doi: 10.1007/s00158-020-02736-8.

[21] L. C. Høghøj, D. R. Nørhave, J. Alexandersen, O. Sigmund, and C. S. Andreasen, "Topology optimization of two fluid heat exchangers," International Journal of Heat and Mass Transfer, vol. 163, p. 120543, Dec. 2020, doi: 10.1016/j.ijheatmasstransfer.2020.120543.

[22] B. S. Mekki, J. Langer, and S. Lynch, "Genetic algorithm based topology optimization of heat exchanger fins used in aerospace applications," International Journal of Heat and Mass Transfer, vol. 170, p. 121002, May 2021, doi: 10.1016/j.ijheatmasstransfer.2021.121002.

[23] A. Della Torre, G. Montenegro, A. Onorati, S. Khadilkar, and R. Icarelli, "Multi-Scale CFD Modeling of Plate Heat Exchangers Including Offset-Strip Fins and Dimple-Type Turbulators for Automotive Applications," Energies, vol. 12, no. 15, p. 2965, Jan. 2019, doi: 10.3390/en12152965.

[24] A. Khan, I. Shah, W. Gul, T. A. Khan, Y. Ali, and S. A. Masood, "Numerical and Experimental Analysis of Shell and Tube Heat Exchanger with Round and Hexagonal Tubes," Energies, vol. 16, no. 2, p. 880, Jan. 2023, doi: 10.3390/en16020880.