Design and Thermal Analysis of Advanced Heat Exchangers Using Computational Methods

B. Santhosh kumar, M Vasista Bharath Kalyan
Siddhartha institute of technology & sciences
santhu.dora@gmail.com; vasistakalyan29@gmail.com

Abstract

Advanced heat exchangers play a vital role in modern thermal systems by enhancing energy efficiency, temperature control, heat recovery, and safety in industrial, automotive, aerospace, electronic cooling, refrigeration, and renewable energy applications. This review article focuses on the design and thermal analysis of advanced heat exchangers based on computational techniques with particular interest on thermal-hydraulic performance, design optimization, and emerging technologies. Various major types of heat exchangers, such as shell and tube, plate, finned-tube, microchannel, spiral, helical, printed circuit, nanofluid-based and additively manufactured heat exchanger designs are discussed in relation to their respective applications and computational needs. Key design parameters like flow arrangement, heat transfer area, flow velocity, Reynolds number, baffles, fins, material properties, and properties of working fluid are discussed to illustrate their effect on heat transfer enhancement and pressure drop. Computational methods are emphasized, including computational fluid dynamics, finite volume method, finite element method, conjugate heat transfer analysis, and optimization techniques, which help to predict the temperature distribution, velocity fields, pressure variation, turbulence behavior, thermal stresses, and performance indicators. The review also highlights recent developments related to compact heat exchangers, microchannel cooling, nanofluids, additively manufactured heat exchangers, artificial intelligence, machine learning, and digital twin monitoring. While computational methods may save experimental costs and help to improve the design, they are not guaranteed to be reliable unless the mesh is high quality, the boundary conditions are realistic, the turbulence model is properly selected, and the model is tested with experiments. In general, the study highlights the need for future development of heat exchangers to be designed with both heat transfer enhancement and hydraulic efficiency, manufacturability, reliability, cost-effectiveness and requirements for the design of the heat exchanger to be service life oriented in a realistic operating condition and long-term service environment.

Keywords: Advanced heat exchangers; thermal analysis; computational fluid dynamics; heat transfer enhancement; pressure drop; nanofluids; numerical simulation; optimization.

1. Introduction

Heat Exchangers are devices used to transfer heat from a higher-temperature fluid to a lower-temperature fluid, or vice versa, without mixing occurring between the two fluids. They are commonly utilized in systems that include heat recovery, conditioning, evaporation, condensation, heating, and cooling in an engineering system. Heat exchangers play a significant role in energy conservation, process efficiency, thermal safety, and environmental sustainability in industrial and domestic applications. They are used in boilers, condensers, evaporators, radiators, intercoolers, oil coolers, solar thermal collectors, geothermal systems, refrigeration plants, nuclear reactors, electronic cooling systems and chemical reactors. The simple function of a heat exchanger is to heat up cold fluid while allowing the hot fluid to cool down. The design of a heat exchanger is complicated since a number of factors, including heat transfer, fluid flow, pressure drop, material selection, fouling, corrosion, manufacturability, compactness, cost, and operational life must be considered simultaneously. If the heat exchanger creates too high of pressure drop, too high of pumping power, vibration, fouling, or difficult maintenance, a high heat transfer rate exchanger is not always the best design. Thus, a thermal-hydraulic optimized heat exchanger design is needed. Empirical correlations, log mean temperature difference method, effectiveness-number of transfer units method and experimental testing have traditionally been used in the design of heat exchangers. These methods can be effective at the preliminary design stage but may not be able to provide detailed information on local temperature distribution, recirculation zones, dead zones, turbulence intensity, wall shear stress, thermal stresses, and local heat transfer coefficient variation. These local effects are significant in many practical systems and have a significant effect on the performance of the heat exchanger. For instance, while poor shell side flow distribution in a shell and tube heat exchanger can decrease effective heat transfer area, too high a flow velocity in microchannels can improve convective heat transfer but also increase pressure drop. Computational approaches have become helpful to address these limitations (Li et al., 2025).

The use of computational fluid dynamics (CFD) for analysis of heat exchangers is now almost common practice as it can be used to calculate the temperature field, velocity profile, pressure distribution, flow maldistribution, fouling sensitive areas etc. and calculate the performance of the heat exchanger for various operating conditions. Previous examples have demonstrated the application of CFD to thermal analysis, prediction of pressure drop, flow maldistribution, assessment of fouling, and optimization of various heat exchanger configurations. In the commercial CFD tools, the turbulent models used in heat exchanger simulation include common turbulence models like standard k–ε, realizable k–ε, RNG k–ε, Reynolds stress model and shear stress transport models. When appropriate meshing, boundary conditions and validation are used, the reported CFD results can be within acceptable agreement with experimental results. The relevance of computational analysis has been further enhanced with the development of new and innovative ideas for heat exchanger technologies. The heat exchanger designs such as compact heat exchanger, micro channel heat exchanger, printed circuit heat exchanger, spiral and helical heat exchanger, corrugated plate heat exchanger, nanofluid-based heat exchanger and additively manufactured heat exchanger have complex geometry and flow patterns, which are hard to be analyzed by simple analytical method. Additive manufacturing, especially, can create complex internal channels, lattices, topology optimized surfaces and compact weight saving heat exchangers, which necessitate an analysis of the numerical data before being produced. The recent reviews suggest that additive manufacturing and artificial intelligence are finding greater potential to be combined for thermal management applications, such as the design, selection of materials, evaluation of hydrodynamic performance, and optimization of structures in thermal management. The aim of this brief review is to present the design and thermal analysis of advanced heat exchangers via computational tools. This review emphasizes the classification of heat exchangers, design parameters, computational models, governing equations, thermal and hydraulic performance indicators, recent advances, limitations and scope for future research (Aslam Bhutta et al., 2012).

The paper aims to offer an academic review of the subject in a succinct but comprehensive manner which can be accessed by undergraduate, postgraduate and early research scholars in the field of thermal engineering, mechanical engineering, energy systems and computational modelling. There are several classes of advanced heat exchangers which are classified according to the construction, arrangement of flows, compactness, technique of heat transfer, and manufacturing process. Even though they are very inefficient, conventional heat exchangers like shell-and-tube, plate and finned-tube are still widely used in industrial applications because of their reliability and design procedures. However, compact size, high heat transfer rate, low pressure drop, and high energy efficiency are increasingly required by modern thermal systems. This has prompted the growth of microchannel, printed circuit, nanofluid based and additively manufactured heat exchangers. In Figure 1, the advanced heat exchangers are clearly classified, and the connection between the conventional and emerging heat exchanger designs is illustrated.

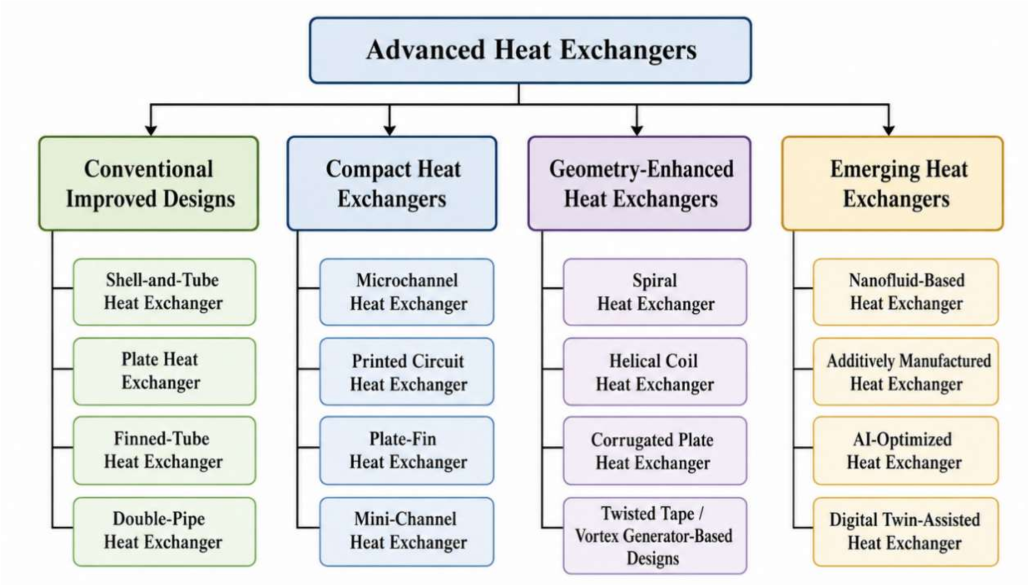


Figure 1: Classification of advanced heat exchangers

Advanced heat exchanger development has gone beyond "traditional" shapes as seen in Figure 1. The shell-and-tube and plate heat exchangers are still significant in large scale industrial systems but the compact and improved design are gaining significance in modern energy, automotive, aerospace and electronic cooling applications. The classification also suggests that computational tools are required since the flow, heat transfer mechanism, and design constraints for each type of heat exchanger are different. Hence, in the remaining sections, the main types of heat exchangers and the computational requirements for analysis are discussed in detail.

2. Classification of Heat Exchangers

Heat exchangers may be categorized according to construction, flow arrangement, heat transfer process, compactness, and application. When considering advanced thermal systems, classification is helpful as each type of heat exchanger offers specific benefits and drawbacks, and the computational modelling needs vary (Zohuri, 2017).

2.1 Shell-and-Tube Heat Exchangers

Shell and tube heat exchangers are widely used in process industries, power plants, refineries, chemical plants and high-pressure systems. They are a collection of tubes which are surrounded by a cylindrical shell. The first fluid is circulated through the tubes and the second fluid flows over the tubes within the shell. Baffle plates are frequently found on the shell side for tube bundle support, turbulence and mixing. The benefits of using shell and tube heat exchangers include that they are robust, can operate at high pressures and temperatures, can be designed flexibly, are easy to maintain and can be used with a variety of fluids. Their performance is significantly influenced by baffle spacing, baffle cut, tube arrangement, shell diameter, tube pitch, fluid velocity and fouling tendency, however. Because shell-side flow has three dimensions and is frequently subject to leakage streams, non-uniform velocity fields, bypass streams and recirculation zones, computational analysis is particularly valuable in shell-and-tube heat exchangers (Raj & Ganne, 2012).

2.2 Plate Heat Exchangers

Plate heat exchangers are made of thin metal plates that are stacked together and have alternating passages for hot and cold fluids. To promote turbulence and more surface for heat transfer, plates are often corrugated. These heat exchangers have a high surface area to volume ratio, are compact, lightweight, and thermally efficient. Food processing, HVAC systems, refrigeration, heat pumps, chemical industries and low to medium pressure thermal systems are common applications for plate heat exchangers. They are affected by the corrugation angle, chevron pattern, plate spacing, gasket arrangement and flow distribution. CFD is useful for analysing local turbulence, wall shear stress, temperature distribution and pressure loss in the narrow-corrugated channels (Zhang et al., 2022).

2.3 Finned-Tube Heat Exchangers

Finned-tube heat exchangers are typically employed when one of the fluids has a low heat transfer coefficient, particularly air. Increased effective heat transfer area is achieved by attaching fins to the tubes. They are commonly employed in air-conditioning units, car radiators, air coolers, condensers, evaporators and electronic cooling. The fin geometry, fin pitch, tube arrangement, air velocity, tube diameter and fin material all affect finned-tube performance. Turbulence and thermal resistance is increased by using wavy fins, louvered fins, perforated fins, and vortex-generator fins. The problem is that there is a tradeoff; with the increase in heat transfer, there can be an increase in pressure drop. Hence, it is important to optimize the selection of a suitable fin design using computational optimization (Yadav et al., 2022).

2.4 Microchannel Heat Exchangers

Microchannel heat exchangers are small-scale heat transfer devices that typically have a hydraulic diameter on the order of micrometers to a few millimeters. They have the ability to remove heat efficiently from a compact system due to their high surface area-to-volume ratio. These are found in electronics cooling systems, car radiators, high-speed computers, fuel cells, space and refrigeration systems. Microchannel systems have high heat transfer coefficients but are susceptible to pressure drop, flow maldistribution, clogging, manufacturing defects and fouling. The most common applications of CFD are to investigate channel shape, channel number, hydraulic diameter, inlet manifold design, heat flux distribution, and coolant selection. Numerical and experimental investigation of microchannel heat exchangers reveals their promising heat flux applications, especially in conjunction with fluid application and optimisation of the geometry (Ramesh et al., 2021).

2.5 Spiral and Helical Heat Exchangers

The spiral and helical heat exchangers are designed with curved flow path to induce secondary flow, mixing and increase the convective heat transfer. Dean vortices are created in helical coil heat exchangers because of the curvature induced centrifugal forces that enhance the mixing of the fluid and thin the boundary layer. The spiral heat exchanger is compact and can be used for viscous fluids, fouling fluids and fluids with self-cleaning flow behaviour. These designs are representative of flow that is strongly 3-D, for which computational analysis can be helpful. The significant parameters are coil diameter, pitch, curvature ratio, Reynolds number, Dean number, fluid properties. CFD can help to identify secondary flow patterns, distribution of wall heat transfer coefficient and pressure loss regions (Sigalotti et al., 2023).

2.6 Additively Manufactured Heat Exchangers

Additively manufactured heat exchangers are a new generation of thermal devices. The use of additive manufacturing makes it possible to create complex internal paths, lattice structures, manifolds and integrated manifolds that cannot be manufactured using traditional manufacturing processes, and topology optimized fins. Studies on AM-based heat exchanger show that this technology can realize compact, lightweight, high efficiency thermal systems, particularly for high-temperature and aerospace/automotive applications. But there are problems associated with additively manufactured heat exchangers like surface roughness, dimensional accuracy, powder removal, material defects, leak tightness, post-processing, and cost. The computational design of the structure can thus be critical to determine the thermal-hydraulic performance and structural reliability prior to manufacture (Sun & Shang, 2022).

The heat exchanger can be in various configurations depending on the application type, operating temperature, pressure range, working fluid, compactness requirement, heat transfer mode. Each type comes with its own computational and design benefit. Plates and microchannel heat exchangers are good for applications requiring compactness and efficiency, while shell-and-tube heat exchangers are better for high pressure industrial systems. Hence it is essential to have a classification of commonly used and advanced types of heat exchangers to understand their applications and how computational analysis can help to improve their performance. Table 1 shows the main types of heat exchanger, their salient features, their common applications and computational interest.

Table 1. Classification of heat exchangers and common applications

Heat exchanger type	Main feature	Common applications	Computational focus	Reference
Shell-and-tube	Robust design with tube bundle inside shell	Power plants, refineries, chemical industries	Shell-side flow, baffle design, pressure drop	(Zhang et al., 2022)
Plate heat exchanger	Corrugated plates with compact arrangement	HVAC, food processing, refrigeration	Plate corrugation, channel flow, local turbulence	(Zohuri, 2017)
Finned-tube	Extended surface area using fins	Air coolers, condensers, radiators	Air-side heat transfer, fin optimization	(Ramesh et al., 2021)
Microchannel	Small hydraulic diameter and compact design	Electronics cooling, automotive cooling	Channel geometry, flow maldistribution, pressure drop	(Sun & Shang, 2022).
Spiral/helical	Curved path causing secondary flow	Viscous fluids, process industries	Dean vortices, mixing, curvature effects	(Yadav et al., 2022)
Printed circuit	Chemically etched compact channels	Nuclear, aerospace, supercritical CO_2 cycles	High-pressure thermal-hydraulic analysis	(Sigalotti et al., 2023)
Additively manufactured	Complex 3D internal passages	Aerospace, compact energy systems	Topology optimization, lattice design, roughness effects	(Aslam Bhutta et al., 2012
Nanofluid-based	Enhanced fluid thermal conductivity	Cooling systems, solar thermal systems	Nanoparticle concentration, viscosity, thermal conductivity	(Li et al., 2025)

Table 1 shows that different heat exchanger types exhibit different thermal and hydraulic behaviour. Shell-and-tube and plate heat exchangers have been in use for years and are reliable and well-known systems. Advanced designs like microchannel, printed circuit, nanofluid-based, and additively manufactured heat exchangers, however, are becoming significant because of their compactness and enhanced heat transfer performance. It is also seen from the table that the computational analysis can not only be applied to predict the heat transfer but also be applied to the estimation of pressure drop, flow distribution analysis, geometry optimization and evaluation of thermal stress.

3. Design Parameters that Impact Thermal Performance

The thermal performance of a heat exchanger is affected by both the geometric and operating parameters. The best design should maximize the heat transfer rate, minimize the pressure drop, pumping power, material cost and maintenance problems (Azeez mohammed Hussein et al., 2022).

3.1 Flow Arrangement

One of the major design considerations is flow arrangement. Heat exchanger configurations can be parallel-flow, counter-flow, cross-flow or mixed-flow. For parallel flow, both fluids are introduced at the same end and flow in the same direction. When the fluids flow in opposite directions it is called Counter flow. The main reason that a counter-flow heat exchanger is usually more thermally effective is it achieves a higher mean temperature difference (MTD) between the fluid streams flowing across the heat transfer surface. Cross-flows are used in compact heat exchangers, radiators, and air coolers. Computational techniques can be used to compare different flow arrangements since they give detailed information about temperature and velocity distributions. The manifold effects, baffles, turns, leakage, bypassing, or maldistribution in practical designs can cause the actual flow pattern to differ from the ideal (Made Arsana & Agung Wahyuono, 2021).

3.2 Heat Transfer Area

The most popular way of enhancing heat exchanger performance is to increase the heat transfer area. This can be accomplished by fins, corrugated plates, extended surfaces, micro channels, porous structures or lattice geometries. However, the larger the surface area the greater the frictional resistance and fouling tendency. Thus, the enhancement of the surface area needs to be assessed in conjunction with the pressure drop. Finned surfaces are particularly effective if the heat transfer is between a liquid and a gas, since convective heat transfer coefficients are smaller for a gas than for a liquid. Plate-fin and tube-fin designs boost the air-side area and enhance heat removal. Microchannels have a very high surface area in a small volume, which is ideal for a miniaturized cooling system in compact heat exchangers (Patil et al., 2022).

3.3 Fluid Velocity and Reynolds Number

The rate of the flow of fluid has an immediate effect on convective heat transfer. The higher the velocity, the more turbulent the flow and the lower the thermal boundary layer thickness, and the higher the heat transfer coefficient. But, higher velocities come with higher pressure drop and higher pumping power. Thus, the design needs to determine an optimum flow rate instead of just increasing the velocity. The Reynolds number helps identify whether the flow is laminar or turbulent or in a transitional state. Turbulent flow tends to have relatively high friction losses as well, but it also has a relatively high heat transfer. Despite the fact that the flow is laminar, high heat transfer can occur in a microchannel because of the small characteristic length and high surface-area-to-volume ratio (Fernandes & Krishanmurthy, 2022).

3.4 Baffles, Inserts, and Vortex Generators

Shell and tube heat exchangers often use baffles as a way to force the flow around the tube bundle and create turbulence on the shell side. Good baffles design will enhance the heat transfer, but too many baffles obstructions will result in high pressure drop and flow induced vibration. The important design parameters are baffle spacing, baffle cut, segmental baffles, helical baffles and inclined baffles. Disturbing the boundary layer with inserts such as twisted tapes, wire coils, ribs and vortex generators to create secondary flow. These enhancement techniques are employed to boost heat transfer, and typically raise the friction factor. So it is necessary to use thermal performance factor or performance evaluation criterion to evaluate their performance rather than heat transfer coefficient (Permatasari & Yusuf, 2018).

3.5 Material Selection

The selection of material has an impact on heat transfer, mechanical strength, corrosion resistance, cost, and durability. The two metals which are widely used in compact heat exchanger and radiators are Copper and Aluminium because of their high thermal conductivity. Strength, corrosion resistance and stainless steel is less conductive. Titanium is used in corrosive applications like marine applications. However, polymers have lower thermal conductivities, so they cannot be used unless the surface area is increased, but they have other advantages where corrosion resistance and low weight are important. The material selection also depends on the printability, the characteristics of the powder, surface finish, post-processing requirements and operating temperature of additively manufactured heat exchangers. While polymer materials are preferred for lower temperature applications, metal Additive Manufacturing is more suitable for high-temperature and high-performance applications due to the thermal conductivity and mechanical resistance properties of the metals (Azeez mohammed Hussein et al., 2022).

3.6 Working Fluid Properties

The performance of heat exchangers is greatly affected by the thermophysical properties of the working fluid. The density, viscosity, thermal conductivity, specific heat, Prandtl number, boiling point, freezing point, corrosion behaviour and stability are important fluid properties. Depending on the application, these are typically water, oils, ethylene glycol, refrigerants, air, molten salts or supercritical CO_2. Nanofluids have garnered interest due to the potential improvement of the effective thermal conductivity of base fluids with the help of nanoparticles. Aside from that, however, there is a potential for viscosity rise, pressure drop, sedimentation, and power consumption when using nanofluids. Studies carried out recently indicate that there is an optimal concentration level for the nanofluid at which the performance may be optimum but further increase in concentration might lead to a decrease in performance due to similar reasons, such as increase in pressure drop or increases in instability ("Review on factors affecting efficiency of shell and tube heat exchanger," 2024). An example of the latter is a nanofluid-enhanced shell-and-tube heat exchanger, which was studied using a computational model in 2026 and found that the pressure drop increased with the concentration of the nanofluid and that the performance evaluation criterion reached an optimum value before dropping. Design and operating conditions greatly affect heat exchanger performance. The overall thermal efficiency and pressure loss of the system depends on the value of the parameters, such as the heat transfer area, velocity of fluid, hydraulic diameter, baffle arrangement, fin geometry, working fluid, material selection, etc. Some changes can enhance heat transfer, but can also result in higher pressure drop, pumping power, manufacturing costs or fouling tendency. Therefore, it is crucial to consider the advantages and disadvantages associated with each design parameter before choosing an appropriate heat exchanger configuration. Table 2 shows the most significant design parameters and their impact on the performance of the heat exchanger.

Table 2. Major design parameters and their influence on performance

Design parameter	**Positive effect**	**Possible limitation**	**Reference**
Increased heat transfer area	Improves heat transfer rate	May increase size, cost, and fouling	(Patil et al., 2022)
Higher fluid velocity	Enhances convection and turbulence	Increases pressure drop and pumping power	(Made Arsana & Agung Wahyuono, 2021)
Smaller hydraulic diameter	Improves compactness and heat transfer	May cause clogging and high pressure drop	(Azeez mohammed Hussein et al., 2022)
Baffles	Improve shell-side mixing	Can increase flow resistance and vibration	(Permatasari & Yusuf, 2018)
Fins	Increase effective heat transfer area	Add weight, manufacturing complexity, and air-side resistance	(Fernandes & Krishanmurthy, 2022)
Corrugations	Increase turbulence and surface area	Increase pressure loss	(Patil et al., 2022).
Nanofluids	Improve thermal conductivity	May increase viscosity and stability concerns	(Made Arsana & Agung Wahyuono, 2021)
Additive manufacturing	Enables complex compact geometries	Surface roughness, cost, and post-processing issues	(Fernandes & Krishanmurthy, 2022)
High-conductivity material	Improves heat transfer	May increase cost or reduce corrosion resistance	(Azeez mohammed Hussein et al., 2022)

Table 2 indicates that there is a compromise between thermal enhancement and hydraulic loss in the design of heat exchanger. Heat transfer can be enhanced by increasing the heat transfer area, by adding fins to the heat transfer surface, by using baffles, or by reducing the size of the channel; these will also increase the flow resistance and the operating costs. Likewise, nanofluids and AM present new avenues of performance enhancement but pose challenges in stability, cost, surface roughness and complexity of fabrication. Hence, computational approaches are necessary to assess these trade-offs before experimental development/industrial implementation.

4. Computational Methods for Heat Exchanger Analysis

The performance of heat exchangers can be predicted using computational methods, prior to their manufacture and testing. All these techniques minimize experimental cost, save design time, enable parametric study, and aid optimization of complex geometry (Abeykoon, 2021).

4.1 Computational Fluid Dynamics

The most popular computational technique used for analysing heat exchangers is computational fluid dynamics (CFD). The governing equations that are solved in CFD are the equations of mass, momentum, and energy conservation and are solved over a discretized domain. A typical CFD workflow includes the development of the heat exchanger geometry with computer-aided design (CAD), the meshing of the fluid and solid domains, the specification of the fluid properties, solid properties, and boundary conditions, the numerical solution of the governing equations, and subsequent analysis in the form of temperature contours, velocity vectors, pressure fields, streamlines, turbulence intensity, and heat transfer coefficients. One of the most important benefits of CFD is the ability to give local information which is not readily available by simple analytical methods. For instance, CFD can be used to locate dead zones, recirculation, uneven flow distribution, hot spots, low heat transfer areas, high pressure loss areas, and fouling prone areas. The design and optimization of plate heat exchanger, shell-and-tube heat exchanger, compact heat exchanger, printed circuit heat exchanger, microchannel heat sink and heat exchanger reactor have all been done using CFD ("Heat transfer and pressure drop in plate fin heat exchangers: A numerical approach," 2024).

4.2 Finite Volume Method

Finite volume is a method that is widely applied in commercially available CFD packages like ANSYS Fluent, ANSYS CFX, STAR-CCM+ and OpenFOAM. The method consists of splitting the computational domain into small control volumes and then applying the conservation equations to each control volume. The finite volume method is generally used for fluid flow and heat transfer since it conserves mass, momentum and energy at the computational level. Finite volume modelling is employed to simulate flow distribution, pressure drop, temperature changes and heat transfer rate in heat exchanger simulations. The quality of the mesh is crucial as it can result in erroneous gradients, numerical diffusion, convergence difficulties, and unreliable answers. To make sure that the results are not sensitive to additional refinement of the mesh, mesh independence studies are required (ResearchHub Nexus Institute, Nigeria et al., 2025).

4.3 Finite Element Method

The finite element method is a very popular method used for structural and thermal stress analysis. For use in heat exchanger applications, the main functions of FEM include the calculation of thermal expansion, stress concentration, vibration, fatigue, deformation and mechanical failure. It is particularly critical in situations of high pressure, high temperature and cyclic thermal loading. If for instance, the materials of the shell and tube are different, then a temperature difference may cause thermal stresses in shell-and-tube heat exchanger. Structural analysis might be needed for additively manufactured heat exchangers due to the presence of thin walls and complex channels that must be able to resist operating pressure, as well as any lattice features. FEM can also be used in conjunction with CFD to carry out fluid-structure interaction or conjugate heat transfer analysis (Teixeira Junior et al., 2024).

4.4 Conjugate Heat Transfer Analysis

Conjugate heat transfer analysis takes into account the fluid flow and heat conduction through solid walls. This is significant because the heat transfer in a heat exchanger is thought to be happening by convection from the hot fluid to the wall, conduction through the wall, and convection from the wall to the cold fluid. In the case of compact heat exchangers, microchannels and systems with complex wall geometry, neglecting wall conduction can result in inaccurate results. The simulations of conjugate heat transfer involve the computational domain comprising the fluid region as well as the solid region. The temperatures and heat fluxes at the fluid-solid boundary are matched. This approach provides more realistic prediction of the wall temperature, R-value, and heat transfer rate (Manna & Chakraborty, 2023).

4.5 Optimization Methods

Often there are conflicting objectives in the design of heat exchangers. Turbulence may help to enhance heat transfer but can also cause an increase in pressure drop. More compactness can help to make the object smaller but more complex to manufacture. Hence, optimized designs are sought using optimization methods. Popular optimization techniques include, but are not limited to, response surface methodology, genetic algorithms, particle swarm optimization, artificial neural networks, multi-objective optimization, topology optimization, or machine learning surrogate models. The use of AI and machine learning for modelling, prediction, condition monitoring and design optimisation of heat exchangers has been growing in the recent literature. The work on machine learning applications to heat exchanger has shown that, with high-quality experimental or numerical data, data-driven tools are capable of helping predict thermal performance, model heat exchanger degradation, and implement condition-based maintenance (Rao et al., 2020). The analysis of a heat exchanger using a computational approach has a systematic procedure, starting with problem definition, and ending with the evaluation of the performance and optimization of the design of the heat exchanger. Computational procedures can enable detailed visualization of fluid flow, temperature distribution, pressure drop, turbulence intensity, and wall heat transfer unlike traditional analytical calculations. To ensure accurate results, the computational workflow needs to be performed correctly, as the geometry creation, mesh generation, boundary conditions, and turbulence model selection can all introduce errors. The general step-by-step computational procedure for the thermal analysis of heat exchanger is shown in Figure 2.

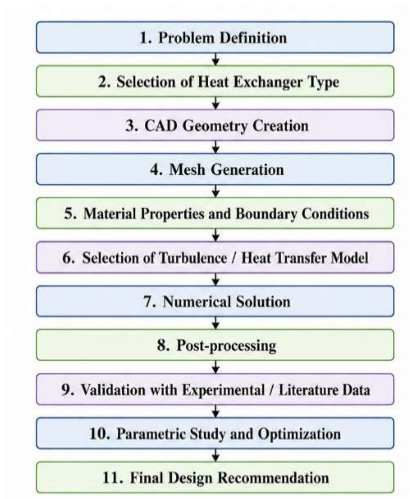


Figure 2. Computational workflow for heat exchanger analysis

From figure 2 it is observed that the computational analysis of the heat exchanger is not confined to the simulation. It is a rigorous task, involving careful planning, geometry preparation, mesh quality checking, choosing the appropriate physical model and assigning suitable boundary conditions. The final results should also be checked with experimental or literature data. Once validated, a computational model can be subject to parametric studies and optimization. This workflow can help minimize the laboratory costs, design time and the prediction accuracy of the heat exchanger performance prior to the manufacturing stage.

Once the computational workflow presented in Figure 2 has been performed, the numerical results need to be translated to measurable performance indicators to evaluate the performance of the heat exchanger designed. The post processing is useful for generating outputs like temperature contours, velocity profiles, pressure fields, wall heat flux and turbulence distribution. But such visual performance is not enough to assess the performance of heat exchanger. They need to be backed up by quantitative characteristics which describe both the thermal behaviour and the hydraulic losses (Naveed & Muhammad, 2024). The thermal performance of the system is indicated by the heat transfer rate, overall heat transfer coefficient, and Nusselt number, while the pressure drop and the friction factor are the hydraulic penalty incurred as fluid flows through the system. Similarly, Reynolds number and Prandtl number represent the flow regime and thermal diffusion characteristics that are needed to compare various operating conditions. Thus, the enhanced heat transfer, alone, should not be the sole objective of a complete computational evaluation. Increases in the heat transfer coefficient can also mean increases in the pressure drop, which can increase pumping power (Wang et al., 2024). Therefore, the thermal and the hydraulic parameters should be analysed simultaneously to determine an efficient and practical heat exchanger configuration. Table 3 is a summary of the major performance parameters that are typically employed in computational heat exchanger studies, and the significance of these parameters in the context of assessing the heat transfer capacity, flow characteristics and overall thermal-hydraulic performance.

There are some thermal and hydraulic parameters that are used to evaluate the performance of a heat exchanger. These parameters allow to quantify the heat transfer capacity, the flow behaviour or the pressure loss and the overall efficiency. For computational studies, these parameters are typically derived from the numerical results including temperature contours, velocity field, pressure distribution and wall heat flux. Reynolds number, Nusselt number, Prandtl number, and friction factor are important dimensionless numbers that can be used for comparisons between various heat exchanger designs and operating conditions. There is some significant performance parameters commonly used in computational heat exchanger analysis, which is presented in Table 3.

Table 3. Important performance parameters in computational heat exchanger studies

Parameter	**Symbol**	**Purpose**
Heat transfer rate	Q	Measures total thermal energy transferred
Overall heat transfer coefficient	U	Indicates total heat transfer ability
Convective heat transfer coefficient	h	Measures convection strength
Nusselt number	Nu	Dimensionless heat transfer indicator
Reynolds number	Re	Identifies flow regime
Prandtl number	Pr	Relates momentum and thermal diffusivity
Friction factor	f	Measures flow resistance
Pressure drop	ΔP	Determines pumping power requirement
Thermal effectiveness	ε	Compares actual and maximum heat transfer
Performance evaluation criterion	PEC	Evaluates heat transfer gain against pressure loss

Table 3 demonstrates that none of the parameters on its own is adequate for judging heat exchanger performance. A high heat transfer rate or Nusselt number would suggest good thermal performance, but must be considered in conjunction with the pressure drop, friction factor and pumping power. In the same way, the thermal effectiveness gives a more general idea of the efficiency of the heat exchanger's utilization of the available temperature difference. Hence, the computational studies should include thermal and hydraulic measurements to give an overall idea of design performance. This integrated assessment is especially crucial in the case of advanced heat exchangers where heat transfer enhancement is accompanied by an increase in flow resistance.

5. Thermal and Flow Performance Analysis

5.1 Temperature Distribution

One of the most critical results of computational thermal analysis is the temperature distribution. The temperature contours indicate the direction of heat transfer from the hot fluid to the cold fluid and can be used to locate hot spots, cold zones and places where heat transfer is light. In a well-designed heat exchanger, temperature changes should be progressive and there should be good heat transfer between the fluids. Temperature distribution is significant with compact heat exchangers and microchannel systems because hot spots can cause damage to electronic components or lower the reliability of the system. CFD simulation can indicate whether or not the coolant is distributed evenly throughout all channels or if some channels are under- or overflowed and cause thermal overload (Yin et al., 2020).

5.2 Velocity Distribution

Understanding flow behaviour with velocity contours and streamlines. The desirable uniform velocity distribution will guarantee the efficient use of the heat transfer surface. Non-uniform velocity may result in over-served areas and under-served areas. CFD can provide information on shell and tube heat exchanger bypass flow, recirculation zones and baffle dead zones. It can demonstrate flow distribution within corrugated channels in plate heat exchangers. In microchannel heat exchangers, it can detect maldistribution due to the poor designed manifold. These insights inform engineers to make changes in geometry prior to fabrication (Ravinthiran et al., 2020).

5.3 Pressure Drop Analysis

A key concern is pressure drop since it affects pumping power. Heat transfer enhancement strategies used in many heat exchanger studies can include increasing turbulence, adding fins, using heat exchanger inserts, or decreasing the channel size. But these methods cause high flow resistance. Thus, a design that has the greatest heat transfer coefficient can not be the most energy-efficient design. The pressure drop analysis is used to determine the feasibility of a heat exchanger to operate in reality. It plays a significant role in Microchannels, Nanofluid systems, and Compact heat exchangers. In the case of nanofluids, higher concentration of nanoparticles can lead to higher thermal conductivity but it can also lead to higher viscosity and pressure drop. A CFD study was recently conducted to study TiO_2-water nanofluids in a double pipe heat exchanger and the results indicated that there was an improvement in convective heat transfer coefficient with increase in Reynolds number and volume fraction of the nanoparticles, but this improvement must be associated with hydraulic effects (Lalagi et al., 2023).

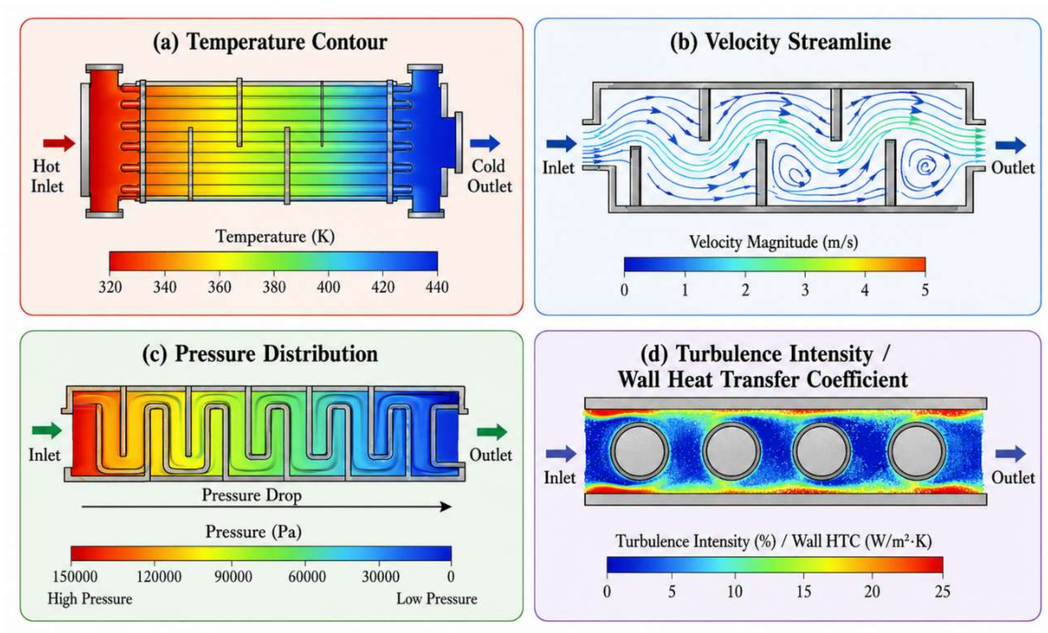
5.4 Heat Transfer Enhancement

The techniques used to improve heat transfer can be broadly categorized as passive, active and compound. Passive methods are those that do not need any outside source of energy and involve things such as fins, ribs, corrugations, twisted tape, rough surfaces, porous media and nanofluids. In the active methods, external energy input is used, such as vibration, pulsating flow, electric field, magnetic field or surface rotation. Compound methods are methods that use two or more enhancement methods. Passive techniques are more commonly employed as they are cheaper and easier. For instance, corrugated plates generate turbulence in plate heat exchangers, twisted tapes generate swirl in tubes, and fins generate more surface area in air-side heat exchangers. It is necessary to consider the thermal-hydraulic metrics for each enhancement approach, however, as the improvements to the heat transfer does not necessarily translate to improvements in overall efficiency (Rathore et al., 2023).

5.5 Fouling and Thermal Degradation

The term fouling is used to describe the unwanted deposition on heat transfer surfaces. It makes the material more thermally insulating, decreases the heat transfer rate, increases pressure drop, and increases maintenance cost. Untreated water, crude oil, biological fluids, or particulate suspensions are typical sources where fouling can be a problem in industrial heat exchangers. The use of computational models can be used to help identify low velocity regions, high temperature zones and recirculation areas where fouling may occur. The use of machine learning and digital twin in fouling prediction and predictive maintenance is also becoming popular. A recent study on machine learning based fouling prediction has revealed that nonlinear data-driven models can help in real-time monitoring of the health of heat exchangers in complex operating conditions. The most important benefit of computational fluid dynamics is that it can output visual results to explain complex thermal and flow behavior within the heat exchanger (Ogunedo, 2020). Typical plots that are generated for the CFD results include temperature contours, velocity streamlines, pressure distribution plot, turbulence intensity contour, and wall heat transfer coefficient map. These outputs can be used to determine the hot spots, low velocity regions, recirculation regions, regions of pressure loss, and regions of poor heat transfer. The typical post processing results of the CFD analysis used to analyze the advanced heat exchanger performance are shown in Figure 3.

Figure 3. CFD post-processing outputs for heat exchanger analysis



CFD post-processing offers detailed information that is not readily available in the traditional design equations as shown in Figure 3. Temperature contours are used for a preliminary assessment of the effectiveness of heat transfer; velocity streamlines show the character of the flow and may indicate a maldistribution. These pressure distribution plots are useful for estimating pumping power requirement and the outputs associated with turbulence are useful for assessing the enhancement of heat transfer. Thus, CFD visualization is a critical tool for understanding the internal behaviour of heat exchangers and is useful for improving its design modification.

6. Recent Advances in Heat Exchanger Design

6.1 Compact Heat Exchangers

The compact heat exchanger is intended to offer high heat transfer area/volume. They find use in systems where space and weight are critical, including in aerospace systems, automotive systems, electronics cooling and portable energy systems. Compacts include plate-fin, microchannel, printed circuit and mini-channel heat exchangers. Compact heat exchangers give a high performance in a small volume, which is the principal advantage. They may however suffer high pressure drop, clogging and fouling because of their small flow passages, and manufacturing difficulty. Optimization of flow passage size, manifold design, heat transfer area, and operating conditions is critical and relies on computational methods (Liao et al., 2021).

6.2 Microchannel and Mini-Channel Designs

Microchannel heat exchangers are now emerging in high heat flux applications. They have a low hydraulic diameter that provides high heat transfer coefficients, and are compact enough to be used in thermal management of electronics and batteries. The microchannel systems performance is influenced by the shape, number of channels, aspect ratio, inlet and outlet manifold design, and properties of the coolant. Numerical simulations can be used in microchannel design due to the difficulty in measuring the conditions within a small channel. CFD can simulate the temperature of the walls, the rise in coolant temperature, the flow distribution, and the pressure drop. An artificial neural network (ANN) has also been used in recent numerical and experimental studies along with microchannel heat exchanger analysis to assist in predicting and optimizing the performance of microchannel heat exchangers (Kasagi et al., 2023).

6.3 Nanofluid-Based Heat Exchangers

Nanofluids are developed fluids which are a mixture of nanoparticles within a base liquid, like water, ethylene glycol or oil. Thermal conductivity and heat transfer properties are enhanced by nanoparticles like Al_2O_3, CuO, TiO_2, ZnO, SiO_2, carbon nanotubes, graphene, and hybrids. Nanofluid based heat exchangers have been noticed as these are capable of enhancing heat transfer while not having to make significant geometric changes. According to the state-of-the-art reviews, nanofluids have raised their potential of heat transfer enhancement, as well as their enhanced thermal properties, which makes them promising working fluids for heat exchangers. But in practice, particle stability, viscosity, corrosion, erosion, pumping power, cost and long term reliability are all factors that need to be taken into consideration. In a computational sense, nanofluids can be modeled either single phase or two phase. The single-phase approach assumes that the nanofluid properties are uniform and the two-phase approach takes into account the slip between the particles and the base fluid. Model choice is dependent on concentration of nanoparticles, particle size, flow regime, desired accuracy (Lalagi et al., 2022).

6.4 Additively Manufactured Heat Exchangers

Additive manufacturing has enabled new opportunities in the design of heat exchangers. Traditional production processes limit the internal structure design due to the limitations of drilling, welding, brazing and cutting. Integrated complex channels, curves passages, lattice structures, surface roughness control and topology optimization design are possible by additive manufacturing. Additive manufacturing has been found to be especially effective in the fabrication of complex shape compact heat exchangers with better thermal performance and lower weight with recent reviews. Fabrication of thin, leakproof walls is still a challenge, as is achieving surface smoothness and eliminating trapped powder, mechanical reliability, and lowering production cost. Because the geometry is too complex for analytical correlations to be used in additive manufacturing-based heat exchanger design, computational methods are necessary. CFD is employed to analyze heat transfer and pressure drop and FEM to analyze mechanical strength and thermal stress. AI optimization can further enhance a design by quickly exploring numerous different geometries (Jafari & Wits, 2018).

6.5 AI and Machine Learning-Assisted Design

AI and machine learning increasingly being applied in heat exchanger research. These methods can simulate the heat transfer coefficient, pressure drop, effectiveness, fouling resistance and optimal operating conditions from experimental or simulation information. Machine learning can also be used as a surrogate model to simplify the repeated CFD simulations. The latest reviews emphasize the critical role of machine learning in revolutionizing heat exchanger modeling, offering precision in prediction, design optimization, and condition monitoring. Usually, common algorithms include artificial neural networks, support vector regression, random forest, gradient boosting, Gaussian process regression, deep learning models. Machine learning models, however, need high quality datasets, feature selection, validation and physical interpretability (Ghalandari et al., 2021).

6.6 Digital Twin-Based Heat Exchanger Monitoring

A digital twin is a virtual model of a physical system that is continually updated and maintained with real-time data. In heat exchanger systems, digital twins may be utilized to keep track of thermal performance, anticipate fouling, forecast the remaining useful life, schedule cleaning cycles, and minimize energy loss. Digital twin based approaches integrate sensor data, physical models, CFD modeling and machine learning. Digital twin applications are in their infancy, but are a significant future application in the world of industrial heat exchanger management. They can move from scheduled maintenance to predictive maintenance, significantly lowering downtime and enhancing energy efficiency (Singh et al., 2021). The modern research about heat exchangers has shifted from the improvement of the conventional design to the new approaches of computing and data-based ones. The analysis of temperature distribution, velocity field, turbulence behavior and pressure drop of heat exchanger in various shapes is very common by using CFD-based studies. Further research directions that are emerging include the study of nanofluids, microchannels, additive manufacturing, artificial intelligence, and digital twin technologies.

The developments and enhancement are targeted towards compactness, thermal efficiency, predictive capability and operational reliability. Table 4 gives a comparative overview of major computational research trends towards advanced heat exchanger design and analysis.

Table 4. Comparative review of computational studies and research trends

Research trend	**Heat exchanger type**	**Computational method**	**Main performance focus**	**Key observation**	**Reference**
CFD-based conventional design	Shell-and-tube, plate, finned-tube	CFD/FVM	Temperature, velocity, pressure drop	Useful for identifying maldistribution and optimizing geometry	(Lalagi et al., 2023)
Baffle optimization	Shell-and-tube	CFD, parametric analysis	Heat transfer and pressure loss	Improved mixing but may increase pumping power	(Jafari & Wits, 2018)
Corrugated surface design	Plate heat exchanger	CFD	Turbulence and local heat transfer	Corrugation improves heat transfer but increases friction	(Ghalandari et al., 2021)
Microchannel cooling	Microchannel heat exchanger	CFD, experimental validation	Hot spot reduction and pressure drop	High heat transfer but sensitive to flow maldistribution	(Singh et al., 2021)
Nanofluid application	Double-pipe, shell-and-tube, microchannel	CFD, single/two-phase modelling	Heat transfer coefficient and PEC	Enhancement depends on concentration and viscosity	(Kasagi et al., 2023).
Additive manufacturing	Compact and lattice heat exchangers	CFD, FEM, topology optimization	Compactness, weight, thermal-hydraulic performance	Enables complex geometry but needs surface roughness control	(Ogunedo, 2020)
AI-assisted modelling	Various heat exchangers	ML, ANN, surrogate models	Fast prediction and optimization	Reduces repeated CFD cost if trained with reliable data	(Rathore et al., 2023).
Digital twin monitoring	Industrial heat exchangers	Physics-informed models and ML	Fouling prediction and maintenance	Supports real-time condition monitoring	(Yin et al., 2020)

From Table 4, it is observed that the heat exchanger computational research has become more and more multi-disciplinary. While traditional CFD studies remain relevant for understanding flow and heat transfer characteristics, newer methods like AI-driven CFD modelling and digital twin monitoring are valuable tools for prediction, optimization, and maintenance. It is also observed here that the advanced heat exchanger technology should not be compared solely in terms of heat transfer enhancement. For practical application, other factors like pressure drop, manufacturability, fouling resistance, prediction accuracy, and real-time monitoring are also crucial.

So, future development of heat exchanger should incorporate CFD, experimental validation, optimization methods and intelligent monitoring systems.

7. Challenges and Limitations

Despite the power of computational methods several challenges need to be considered for reliable analysis of heat exchangers. The CFD results are sensitive to the mesh quality. A coarse mesh will not produce boundary layers, recirculation zones, and temperature gradients; a very fine mesh will require more computational time. To ensure the solution is grid size independent, mesh independence studies are required. For turbulent flow and conjugate heat transfer simulations near wall mesh refinement is particularly important. One of the primary areas of uncertainty in CFD is turbulence modelling. Various turbulence models can give different results with regard to separation, recirculation, swirl and wall heat transfer. Commonly used, robust standard k-ε models, but may not capture complex separated flows. SST k–ω models can be a better choice near the walls, and Reynolds stress models can give a more detailed representation of the turbulence but can be more demanding in terms of computational effort. Realistic boundary conditions are key to simulation accuracy. All inlet temperature, mass flow rate, heat flux, outlet pressure and wall roughness should be consistent with the actual operating conditions. Extraneous assumptions can cause false conclusions. For instance, if the inlet flow velocity is not uniform, assuming it is uniform may overestimate the performance of the heat exchanger. The results of the computation should be checked by experimental or literature data of good quality. Until validated, the results of CFD are only a numerical prediction. The validation may be done by measuring outlet temperature, pressure drop, heat transfer coefficient, Nusselt number or thermal effectiveness. The previous CFD reviews highlight that validated CFD simulations can yield a good level of prediction accuracy and remove the need for expensive prototypes.

There are a number of challenging phenomena, such as fouling, boiling, condensation, particle deposition, phase change, and chemical reactions, that cannot be accurately modeled. The phenomena are characterized by time dependent processes, by complex interfaces and by uncertain material behaviour. There are many models that do not completely represent actual industrial conditions. One assumption for the use of nanofluids in simulations is that the dispersion of particles in the fluid is stable and uniform. In reality, the nanoparticles can clump together, sink, bind to surfaces or become thicker with time. So, one should be cautious with computational predictions of the nanofluid performance and validate them with experiments. The challenges associated with additive manufacturing of HEs include surface roughness, powder removal, dimensional deviations, porosity, residual stresses and post-processing. These manufacturing effects may have significant pressure drop and heat transfer changes. To conclude, idealized CAD-based CFD models are only valid when printed parts are manufactured without defects and roughness, which should be integrated into the model. When creating high fidelity simulations of complex heat exchangers, a lot of computational resources are needed. Long simulation time may be needed for large models with turbulent flow, conjugate heat transfer, multiphase flow or transient analysis. Use of machine learning surrogate models can decrease cost, but demands good training data.

While computational methods give detailed information on heat exchanger performance, there are a number of modelling factors that influence the accuracy of the computed performance. Numerical predictions are susceptible to the quality of the mesh, the selection of the turbulence model, the choice of the boundary conditions, the material properties and the validation data. Furthermore, complex heat exchanger systems can be developed using nanofluids, fouling, multiphase flow and additive manufacturing, which add additional modelling complexity. It is therefore, essential to list the main problems affecting the computational analysis of a heat exchanger and recommend appropriate remedies. Summary of key challenges and possible solutions for computational heat exchanger studies is summarized in Table 5.

Table 5. Challenges and possible solutions in computational heat exchanger analysis

Challenge	Effect on analysis	Possible solution	Reference
Poor mesh quality	Inaccurate gradients and convergence issues	Mesh independence test and boundary layer refinement	(Lal & Li, 2014)

Wrong turbulence model	Incorrect prediction of separation and heat transfer	Compare turbulence models with validation data	(F. Gao et al., 2017)
Unrealistic boundary conditions	Misleading thermal performance	Use experimental or operating data	(Addy et al., 2025)
Lack of validation	Low reliability of results	Validate with literature or laboratory results	(Hampton, 2025)
Complex fouling behaviour	Difficult long-term prediction	Use coupled CFD, empirical fouling models, and digital twins	(Han et al., 2019)
Nanofluid instability	Overprediction of heat transfer	Include viscosity, stability, and concentration effects	(Alvarez, 2023)
AM surface roughness	Higher real pressure drop than simulation	Include roughness models and printed-part measurements	(Salehi et al., 2022)
High computational cost	Time-consuming optimization	Use surrogate models and AI-based optimization	(Wright et al., 2023)

Table 5 indicates that computational analysis is very useful but should be carried out with care to avoid incorrect and misleading conclusions. To ensure reliable simulation results, proper mesh independence testing, realistic boundary conditions, turbulence model comparison, and experimental validation are essential. It can be seen that emerging technologies like nanofluids and additively manufactured heat exchangers also need attention, where idealized simulations may not be able to capture real operating conditions. Hence, future research should integrate the numerical modeling with experimental verification, uncertainty analysis, and practical manufacturing aspects to enhance the reliability of the advanced heat exchanger designs.

8. Future Research Scope

Advanced heat exchangers should be developed to attain high thermal performance at low pressure drop, low material consumption, compact, low impact on the environment, and high operational reliability in the future. Firstly, there is a need for more experimental validation of CFD studies. Numerous numerical studies have been reported with an improvement in heat transfer with not enough experimental confirmation. Validation should comprise of pressure drop, outlet temperature, thermal effectiveness and uncertainty study. Reporting of mesh independence and errors due to the turbulence model is to be clearly communicated. Secondly, it is recommended that future designs should focus on the thermal-hydraulic optimization rather than heat transfer enhancement alone. The useful design should use the best combination of Nusselt number improvement, friction factor and pumping power. All enhanced designs should be evaluated fairly using the performance evaluation criteria. Third, the use of AM for small and complex heat exchanger should be investigated. In future, consideration of real manufacturing factors including surface roughness, powder removal, minimum wall thickness, leakage, residual stress and post-processing should be considered. Topology optimization and AM processes can be used together to develop lightweight and efficient heat exchangers. Fourthly, there is a need for long-term studies of the stability, corrosion, erosion, viscosity, cost and environmental effects of nanofluid-based heat exchanger. Based on the discussion above, one can conclude that hybrid nanofluids and carbon based nanofluids give better thermal performance, but their performance in practical applications should be tested in real operating conditions. Fifth, contribute the use of artificial intelligence and machine learning to the CFD and experimental data. AI models can speed up design optimization, forecast the performance of heat transfer, identify fouling and aid predictive maintenance. Future studies on AI should, however, be directed towards explainability, physical consistency and transferability to other types of heat exchangers. Last, a digital twin-based heat exchanger monitoring system must be designed for industrial systems. Digital twins help to predict fouling, optimize cleaning schedules and minimize energy loss by integrating real-time sensor data, thermal models, CFD-based correction factors, and machine learning. Last but not least sustainability shall be a major design goal. Recyclable materials, low-carbon manufacturing routes, integration of waste heat recovery and energy-efficient operation should be applied to future heat exchangers. Computational techniques can be used as a tool for sustainability that minimizes trial and error experimentation and optimizes the use of resources during the design process.

9. Conclusion

The advanced heat exchanger is an important component for enhancing energy efficiency, thermal management and sustainability in industrial, automotive, aerospace, electronic cooling, refrigeration and renewable energy applications. This review emphasizes the fact that the enhancement of heat transfer is not the only criterion used for the performance of modern heat exchangers, but also the pressure drop, the pumping power, material properties, compactness, fouling resistance, manufacturing methods, and reliability. Conventional analytical methods may not be enough to understand the temperature distribution, velocity fields, pressure variation, turbulence behaviour and thermal stresses involved; computational methods, such as CFD, finite volume modelling, finite element analysis and conjugate heat transfer simulation, offer detailed insight. The research also reveals that enhanced heat exchanger designs like microchannel, printed circuit, nanofluid, spiral, helical and additively manufactured heat exchangers have great potential in the development of compact and high-performance thermal systems. The challenges that come with such benefits are pressure loss, sensitivity of the mesh, uncertainty of the turbulence model, instability of the nanofluid, fouling, surface roughness, and experimental validation. Further tools that are emerging, including Artificial Intelligence, Machine Learning, Surrogate modelling, Topology optimization and digital twins, can help to further speed up design prediction, monitoring, maintenance and optimization. In conclusion, the development of a reliable heat exchanger involves a comprehensive approach that encompasses validated computational modelling, experimental validation, thermal-hydraulic performance assessment, practical manufacturing aspects, and sustainability-driven design. Further studies should therefore be directed to the development of balanced, energy-efficient, cost-effective and environmentally friendly heat exchanger solutions for advanced engineering application scenarios in realistic operating and service conditions.

References

1. Addy, M. N., Aigbavboa, C., & Kwofie, T. E. (2025). Thermal comfort. In *Green Energy and Technology* (pp. 49–71). Springer Nature Switzerland.
2. Alvarez, A. N. T. (2023). *Liquid Viscosity*. Unpublished. https://doi.org/10.13140/RG.2.2.26538.84163
3. Azevedo, B. F., Rocha, A. M. A. C., & Pereira, A. I. (2024). Hybrid approaches to optimization and machine learning methods: a systematic literature review. *Machine Learning*, *113*(7), 4055–4097. https://doi.org/10.1007/s10994-023-06467-x
4. Bhuvaneshwarri, M. Maheswari, C. Kalaivanan, P. Deepthi, Kanth, T. V. R., & V. Saravanan. (2025). Hybrid Swarm Intelligence-Based Neural Framework for optimizing real-time computational models in engineering systems. *International Journal of Computational and Experimental Science and Engineering*, *11*(1). https://doi.org/10.22399/ijcesen.1001
5. Dillon, O., Lau, B., Vinod, S. K., Keall, P. J., Reynolds, T., Sonke, J.-J., & O'Brien, R. T. (2025). Real-time spatiotemporal optimization during imaging. *Communications Engineering*, *4*(1), 61. https://doi.org/10.1038/s44172-025-00391-9
6. Duan, F., Han, B., & Bu, X. (2025). Synergistic integration of refined pelican optimization algorithm and deep neural networks for autonomous vehicle control in edge computing architectures. *Scientific Reports*, *15*(1), 19338. https://doi.org/10.1038/s41598-025-98486-y
7. Fotopoulos, G. B., Popovich, P., & Papadopoulos, N. H. (2024). Review non-convex optimization method for machine learning. In *arXiv [cs.LG]*. https://doi.org/10.48550/ARXIV.2410.02017
8. Gambella, C., Ghaddar, B., & Naoum-Sawaya, J. (2021). Optimization problems for machine learning: A survey. *European Journal of Operational Research*, *290*(3), 807–828. https://doi.org/10.1016/j.ejor.2020.08.045
9. Gao, F., Wang, H., & Wang, H. (2017). Comparison of different turbulence models in simulating unsteady flow. *Procedia Engineering*, *205*, 3970–3977. https://doi.org/10.1016/j.proeng.2017.09.856
10. Gao, M. (2016). Nanomaterials for water treatment: Removal of heavy metals and organic pollutants. *Environmental Science and Pollution Research*, *23*(2), 941–955.
11. Ghalandari, M., Irandoost Shahrestani, M., Maleki, A., Safdari Shadloo, M., & El Haj Assad, M. (2021). Applications of intelligent methods in various types of heat exchangers: a review. *Journal of Thermal Analysis and Calorimetry*, *145*(4), 1837–1848. https://doi.org/10.1007/s10973-020-10425-3
12. Guler, A. U., Demirović, E., Chan, J., Bailey, J., Leckie, C., & Stuckey, P. J. (2022). A divide and conquer algorithm for predict+optimize with non-convex problems. *Proceedings of the AAAI Conference on Artificial Intelligence*, *36*(4), 3749–3757. https://doi.org/10.1609/aaai.v36i4.20289

13. Hampton, T. (2025). From the literature. *Circulation*, *152*(15), 1113–1114. https://doi.org/10.1161/circulationaha.125.077383
14. Han, Z., Xu, Z., & Yu, X. (2019). CFD modeling for prediction of particulate fouling of heat transfer surface in turbulent flow. *International Journal of Heat and Mass Transfer*, *144*(118428), 118428. https://doi.org/10.1016/j.ijheatmasstransfer.2019.07.078
15. Itzhaky, S., Singh, R., Solar-Lezama, A., Yessenov, K., Lu, Y., Leiserson, C., & Chowdhury, R. (2016). Deriving divide-and-conquer dynamic programming algorithms using solver-aided transformations. *Proceedings of the 2016 ACM SIGPLAN International Conference on Object-Oriented Programming, Systems, Languages, and Applications*, 145–164.
16. Jafari, D., & Wits, W. W. (2018). The utilization of selective laser melting technology on heat transfer devices for thermal energy conversion applications: A review. *Renewable and Sustainable Energy Reviews*, *91*, 420–442. https://doi.org/10.1016/j.rser.2018.03.109
17. Ji, R., Zhao, Y., Xiong, Y., Wang, D., Zhang, L., & Hu, Z. (2024). Decomposition-based synthesis for applying divide-and-conquer-like algorithmic paradigms. *ACM Transactions on Programming Languages and Systems*, *46*(2), 1–59. https://doi.org/10.1145/3648440
18. Kareem, S. W., Hama Ali, K. W., Askar, S., Xoshaba, F. S., & Hawezi, R. (2022). Metaheuristic algorithms in optimization and its application: a review. *JAREE (Journal on Advanced Research in Electrical Engineering)*, *6*(1). https://doi.org/10.12962/jaree.v6i1.216
19. Kasagi, N., Suzuki, Y., Shikazono, N., & Oku, T. (2023). Optimal design and assessment of high performance micro bare-tube heat exchangers. *Proceeding of Compact Heat Exchangers and Enhancement Technology for the Process Industries - 2003*, 241–246.
20. Kozhubaev, Y., Novak, D., Ershov, R., Xu, W., & Cheng, H. (2025). Research on navigation and dynamic symmetrical path planning methods for automated rescue robots in coal mines. *Symmetry*, *17*(6), 875. https://doi.org/10.3390/sym17060875
21. Lal, R., & Li, Z. (2014). *Sensitivity analysis of a mesh refinement method using the numerical solutions of 2-D driven cavity flow*.
22. Lalagi, G., Nagaraj, P. B., Veerabhadrappa Bidari, M., & Hegde, R. N. (2022). Influence of design of microchannel heat exchangers and use of nanofluids to improve the heat Transfer and Pressure drop characteristics: A review. *International Journal of Ambient Energy*, *43*(1), 6849–6877. https://doi.org/10.1080/01430750.2022.2056918
23. Li, M., Jiang, Y., Zhang, Y., & Zhu, H. (2023). Medical image analysis using deep learning algorithms. *Frontiers in Public Health*, *11*, 1273253. https://doi.org/10.3389/fpubh.2023.1273253
24. Liao, G., Li, Z., Zhang, F., Liu, L., & Jiaqiang. (2021). A review on the thermal-hydraulic performance and optimization of compact heat exchangers. *Energies*, *14*(19), 6056. https://doi.org/10.3390/en14196056
25. Liu, X., Qi, H., Jia, S., Guo, Y., & Liu, Y. (2025). Recent advances in optimization methods for machine learning: A systematic review. *Mathematics*, *13*(13), 2210. https://doi.org/10.3390/math13132210
26. Mailund, T. (2021). Divide and conquer and dynamic programming. In *Introduction to Computational Thinking* (pp. 247–288). Apress.
27. Myllis, G., Tsimpiris, A., Aggelopoulos, S., & Vrana, V. G. (2025). High-performance computing and parallel algorithms for urban water demand forecasting. *Algorithms*, *18*(4), 182. https://doi.org/10.3390/a18040182
28. Naayini, P., & Kamatala, S. (2023). High-performance data computing: Parallel frameworks, execution strategies, and real-world deployments. *International Journal Of Scientific Advances*, *4*(6). https://doi.org/10.51542/ijscia.v4i6.33
29. Olu Olagunju, S., Folorunsho Akinmuyise, M., & Daniel Ogunbona, B. (2025). Numerical solution of optimization problems via multiplier method. *Ajayi Crowther Journal of Pure and Applied Sciences*, *4*(2). https://doi.org/10.56534/acjpas.v4i2.158
30. Philip, B. V., Alpcan, T., Jin, J., & Palaniswami, M. (2019). Distributed real-time IoT for autonomous vehicles. *IEEE Transactions on Industrial Informatics*, *15*(2), 1131–1140. https://doi.org/10.1109/tii.2018.2877217
31. Ravinthiran, A., Babu, T., Raj, G. T., Siddaarth, P. Y., & Kamatchi, K. S. (2020). Flow variations and comparisons of heat exchanger headers with different cross sections. *PROCEEDINGS OF INTERNATIONAL CONFERENCE ON RECENT TRENDS IN MECHANICAL AND MATERIALS ENGINEERING: ICRTMME 2019*.
32. Rebentrost, P., Schuld, M., Wossnig, L., Petruccione, F., & Lloyd, S. (2019). Quantum gradient descent and Newton's method for constrained polynomial optimization. *New Journal of Physics*, *21*(7), 073023. https://doi.org/10.1088/1367-2630/ab2a9e
33. Salehi, H., Cummins, J., Gallino, E., Harrison, N., Hassanpour, A., & Bradley, M. (2022). A new approach to quantify powder's bed surface roughness in additive manufacturing. *Powder Technology*, *407*(117614), 117614. https://doi.org/10.1016/j.powtec.2022.117614
34. Shi, Y., Lian, L., Shi, Y., Wang, Z., Zhou, Y., Fu, L., Bai, L., Zhang, J., & Zhang, W. (2023). Machine learning for large-scale optimization in 6G wireless networks. *IEEE Communications Surveys & Tutorials*, *25*(4), 2088–2132. https://doi.org/10.1109/comst.2023.3300664

35. Singh, H., Mishra, U., Saxena, P., Shetiya, G., & Puri, Y. M. (2021). Digital twin for shell and tube heat exchanger in industry 4.0. In *Advances in Mechanical Engineering* (pp. 637–644). Springer Singapore.
36. Sun, S., Cao, Z., Zhu, H., & Zhao, J. (2020). A survey of optimization methods from a machine learning perspective. *IEEE Transactions on Cybernetics*, *50*(8), 3668–3681. https://doi.org/10.1109/TCYB.2019.2950779
37. Wright, L. M., Liu, J., Kamrath, B., & Morris, M. C. (2023). Evaluation of additively manufactured heat exchanger tubes using experimental and conjugate heat transfer analysis. *Volume 7B: Heat Transfer — General Interest/Additive Manufacturing Impacts on Heat Transfer; Internal Air Systems; Internal Cooling*.
38. Yin, L., Wang, M., Zhou, J., & Li, T. (2020). Design and numerical simulation of a new microchannel heat sink. In *Lecture Notes in Electrical Engineering* (pp. 334–344). Springer Singapore.