OPEN ACCESS
Global Journal of Research in Engineering & Computer Sciences
ISSN: 2583-2727 (Online)
Volume 06 | Issue 01 | Jan.-Feb. | 2026
Journal homepage: https://gjrpublication.com/gjrecs/
Research Article

# Enhancing Heat Exchanger Efficiency Using Computational Fluid Dynamics (CFD)Techniques

*Karrar A. Alakoul

Department of Electronic and Communications, College of Engineering, Al-Muthanna University.

**DOI:** 10.5281/zenodo.18217746 **Submission Date:** 22 Nov. 2025 | **Published Date:** 12 Jan. 2026

***Abstract***

*The aim of the present study was to assist in the development of increased thermal productivity of shell-and-tube type heat exchangers through computational fluid dynamics analysis along with design optimization of heat exchangers. Simulations were run for various Reynolds numbers between 5,000 and 30,000, using the standard k–ε turbulence model to evaluate how different conditions of flow and structural configurations affect a heat exchanger's overall performance. In addition, baffle spacing ratios and turbulence-inducing elements were tested to examine the effects of these parameters on heat transfer and pressure drop within the shell-and-tube heat exchangers. It was concluded that both increasing the number of baffles spacings along with the introduction of flow-disturbing elements would increase the convective heat transfer capabilities of the heat exchangers. Specifically, the Nusselt number and heat transfer coefficient were significantly increased due to the combination of the reduced baffle spacings and the controlled experimental flow disturbance, although frictional losses were also increased to a moderate degree. A baffle spacing of 0.3 times the shell diameter was found to have the best performance overall, producing approximately 10% greater thermal effectiveness compared to unmodified or poorly-baffled heat exchangers without incurring large hydraulic or legal penalties.*

***Keywords:*** *Computational Fluid Dynamics (CFD), Shell-and-Tube Heat Exchanger, Thermal Performance Optimization, Baffle Spacing Ratio.*

## 1. Introduction

Heat exchangers are important for improving the efficiency of energy use, as well as for addressing the energy challenges faced by many different types of industries [1,2]. They are devices that enable the transfer of thermal energy from one fluid to another fluid at two different temperature levels. This occurs in many applications such as HVAC, Power Generation, Refrigeration, Automotive Cooling, Chemical Processing and Food Processing [[3], [4], [5]]. Effective management of thermal energy will reduce the amount of energy used and the impact on the environment. This has consistently driven innovation in the design and technology associated with heat exchangers. There are many different types of heat exchangers; one of the most well-known are helical coil tubes. Helical coil tubes are unique because of their compact design, low-pressure drops, and the ability to transfer greater amounts of heat than straight tube systems [6,7]. The unique shape of a helical coil tube enables the most efficient use of the heat transfer surface area within a specific volume, while simultaneously improving flow dynamics through the creation of Dean vortices. The creation of secondary flows allows for improved heat transfer efficiency through disruption of the thermal boundary and enhancement of the mixing between the main flow and the fluid near the tube's interior wall [8,9]. In addition to improving heat transfer efficiency, due to their unique structure, helical coils are also less susceptible to fouling. This means that heat exchangers designed with helical coils require less maintenance and provide prolonged operational efficiency which is critical in many industrial applications, where long-term reliability is necessary [10,11].

In heat transfer, there are two main factors (temperature and heat flux) that have a major impact on thermal system performance (heat exchangers) — as a substance gains temperature, the thermal instability of its constituent parts increases, causing an increase in the complexity of the entire system's thermal characteristics [4]. Conduction is one of three core methods of heat transfer, wherein heat transfers from an area of a higher temperature toward an area of a lower temperature. This transfer process may occur in solids, liquids, or gases, even at material interface surfaces that are directly adjacent to one another [5]. On the contrary, convection is the principal method of energy transfer between a solid and a fluid, while motion of the fluid is ongoing; therefore, convection plays an essential role in CFD simulations. The temperature gradient that occurs within the fluid surrounding the solid surface results in density differences in the fluid, which generates the fluid flow cycles. One such example of forced convection is air circulating over fins in a heat exchanger (an effect that CFD simulations commonly consider); as a result, there is an increase in heat transfer rate. Finally, heat transfer can happen through radiation between two masses of matter, even at vast distances apart, if a heat difference exists between both [6]. As the temperature of the fluid in a heat exchanger tube is increased or decreased, there will also be substantial alterations in the flow properties of the liquid. The changes include a reduction in the viscosity of the liquid and its subsequent phase transition, the possibility of cavitation, freezing or melting of the liquid, and marked changes in its density. In the case of a two-phase flow (gas-liquid), temperature is a critical factor influencing the manner in which the fluid flows. When heat is transferred between the two phases, it is accomplished through three mechanisms: conduction, convection, or radiation. Numerical modeling of these mechanisms accurately represents the flow phenomena and is thus a primary goal of CFD simulations [7]. This chapter reviews the different correlations for the Nusselt number and friction factor for various operating conditions, and also discusses how nanofluids can enhance heat exchanger performance through the synergistic effects of their use. Fourier, the originator of heat conduction theory, was the first to mathematically model the thermal behavior of matter in a bulk medium (i.e., not in discrete elements). His initial study provided the foundation for the governing equations used today in the numerical approaches used in CFD [8][9]. Nanofluids contain suspended nanoparticles that have better thermal properties than traditional fluids and are therefore being utilized as advanced heat transfer agents and incorporated into CFD simulations to further enhance the performance of heat exchangers [10][11]. With the increasing demand for high heat flux operations, advanced heat transfer techniques are rapidly gaining attention. This includes the use of nanofluids in cooling systems for industrial equipment, automotive engines, welding equipment, etc. [12]. Nanomaterials and nanoparticles have been produced for many years and are widely used today for making nanofluids for Thermal Applications in Heat Exchangers. To achieve successful industrial applications and CFD modeling, it is necessary to use controlled, narrowly sized distribution of particles. Different types of synthesis technology are used to make stable nanofluids of Nanoparticles with controlled particle size, including Milling, Gas Phase, and Liquid Phase Process [13].

## 2. Theoretical Foundations

### 2.1. Fundamentals of Heat Transfer

Heat transfer is one of the most basic physical processes in engineering systems. It is very important for designing thermal equipment, such as heat exchangers. The three main mechanisms of heat transfer are conduction, convection, and radiation. In heat exchangers, conduction and convection are the dominant mechanisms [12].

The law of heat conduction is expressed as:

$$q = -k\frac{dT}{dx}$$

where:
q — heat flux (W/m²)
k — thermal conductivity of the material (W/m·K)
dTdx — temperature gradient along the direction of heat transfer
For steady-state, one-dimensional conduction between two surfaces with temperatures T1 and T2, the total heat transfer rate is given by:

$$Q = kA\frac{(T_1 - T_2)}{L}$$

In convection, heat transfer occurs along with the bulk motion of the fluid and follows:

$$Q = hA(T_s - T_\infty)$$

where h is the convective heat transfer coefficient, which depends on the properties of the fluid [6], flow velocity, and flow regime (laminar or turbulent).

The total heat transfer in a heat exchanger is expressed by [18]:

$$Q = UA\Delta T_m$$

where:
U: overall heat transfer coefficient (W/m²·K)
A: effective heat transfer area (m²)
ΔTm\Delta T_mΔTm: logarithmic mean temperature difference (LMTD), calculated by:

$$\Delta T_m = \frac{(T_{h,i} - T_{c,o}) - (T_{h,o} - T_{c,i})}{\ln \frac{(T_{h,i} - T_{c,o})}{(T_{h,o} - T_{c,i})}}$$

## 2.2. Heat Exchangers and Their Types

A heat exchanger is a device designed to transfer thermal energy between two or more fluids at different temperatures without allowing them to mix. Based on structure and flow arrangement, heat exchangers are classified into several types:

1. Shell-and-tube heat exchangers
2. Plate heat exchangers
3. Compact heat exchangers
4. Coiled or double-coil heat exchangers

In shell-and-tube exchangers, one fluid flows through the tubes while another flows around them within the shell. Parameters such as tube diameter, baffle spacing, number of passes, and flow arrangement (parallel, counter, or cross-flow) directly affect heat transfer rates and pressure drops. Increasing the heat transfer area or using fins and controlled surface roughness can enhance the overall coefficient UUU, but usually at the cost of higher-pressure losses.

## 2.3. Principles of Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that solves the governing equations of fluid motion and heat transfer numerically. It is based on three fundamental equations:

1. Continuity equation (mass conservation):

$$\nabla \cdot (\rho \vec{V}) = 0$$

2. Momentum equation (Navier–Stokes):

$$\rho \left( \frac{\partial \vec{V}}{\partial t} + (\vec{V} \cdot \nabla)\vec{V} \right) = -\nabla P + \mu \nabla^2 \vec{V} + \rho \vec{g}$$

3. Energy equation:

$$\rho C_p \left( \frac{\partial T}{\partial t} + \vec{V} \cdot \nabla T \right) = k \nabla^2 T + S$$

In CFD, these equations are discretized using methods such as the Finite Volume Method (FVM), Finite Difference Method (FDM), or Finite Element Method (FEM) and solved over a computational grid. One of the most critical steps in CFD is turbulence modeling, since most industrial flows are turbulent. Common turbulence models include:

### 2.4. Application of CFD in Heat Exchanger Analysis

Using CFD allows scientists and engineers to study and understand how fluids move around surfaces, how fluids change temperature and how fluids transfer heat from one part of their system to another. On top of that, CFD simulates physical effects (such as vortexes, vortices and flow separation) that cannot be observed in the laboratory or simulated in the vacuum of space [5].

The Nusselt number is a major factor when evaluating heat exchanger performance. It describes the effect that convection has over pure conduction by providing an index of how much increased heat transfer through a heat exchanger through convection versus through pure conduction.

$$Nu = \frac{hD}{k}$$

$$Re = \frac{\rho V D}{\mu}$$

$$Pr = \frac{C_p \mu}{k}$$

The Overall analysis of these dimensionless numbers allows for the prediction of flow characteristics, turbulence intensity, and heat transfer characteristics. In CFD studies, the validation of predictions against Nusselt number and friction factor experimental measurements is typically conducted to ensure the prediction model is a true representation of the real world described by the CFD predictions [14].

## 3. Methodology

In this work, a computational and analytical approach was used in order to study and improve the thermal efficiency of heat exchangers via Computational Fluid Dynamics (CFD). In performing CFD simulations, various design and operating conditions were tested to determine the effects of a variety of input variables on thermal performance in order to identify the best overall design for maximizing heat transfer while minimizing pressure drop. The simulation processes are divided into five (5) stages: creating geometry; creating meshes; defining boundary conditions; setting up numerical solutions; and validating and analyzing results.

To provide input to CFD simulations, 3D models of the heat exchangers were created using design software tools such as SolidWorks, ANSYS Design Modeler, etc. Shell-and-tube configurations are commonly used in heat exchangers because they have been adequately documented and appear to be a proven design. In creating a 3D model using CAD, the model represents the overall shell design and includes the internal tubes, fluid inlet and outlet connections, as well as baffles that are used to promote fluid mixing and increase turbulence.

### 3.1. Mesh Generation

The mesh generation process is the process of breaking a physical domain (workpiece and tools) into smaller subdomains (elements), this makes it easier to obtain a numerical solution to the PDEs that describe the behaviour of a metal forming system.

The majority of applications for the process of mesh generation are on the finite-element analysis; however, it is also used in other ways for example in animation type movies and video games using computer graphical methods.

### 3.2. Governing Equations and Numerical Model

• Continuity equation:

$$\nabla \cdot (\rho \vec{V}) = 0$$

• Momentum equation:

$$\rho\left(\frac{\partial \vec{V}}{\partial t}+\vec{V}\cdot\nabla\vec{V}\right)=-\nabla P+\mu\nabla^2\vec{V}$$

• Energy equation:

$$\rho C_p\left(\frac{\partial T}{\partial t}+\vec{V}\cdot\nabla T\right)=k\nabla^2 T$$

To simulate turbulent flow:

$$\frac{\partial(\rho k)}{\partial t}+\nabla\cdot(\rho k\vec{V})=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right]+G_k-\rho\varepsilon$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla\cdot(\rho\varepsilon\vec{V})=\nabla\cdot\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right]+C_{1\varepsilon}\frac{\varepsilon}{k}G_k-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

## 4. Validation of the CFD Model

By using geometry, users are able to specify their preferred thermal performance and hydraulic characteristics for their own heat exchanger. Selecting a tube diameter of 12.7mm is typical in industrial applications; therefore, it provides a good compromise between heat transfer area and pressure drop. By selecting the housing diameter and number of tubes, users provide enough cross-sectional area for proper distribution of fluids while still maintaining a compact configuration. With Baffle Space Ratio (BSR) ranging from $0.5D_s$ to $0.3D_s$ users can determine how the redirection of flow affects both turbulence intensity and shell-side mixing. In order for users to maximize their overall heat transfer efficiency, they should select the counter flow arrangement as this maximizes the user's logarithmic mean temperature difference (LMTD).

$$Nu=0.023Re^{0.8}Pr^{0.3}$$

$$Nu=\frac{(f/8)(Re-1000)Pr}{1+12.7(f/8)^{1/2}(Pr^{2/3}-1)}$$

$$PEC=\frac{(Nu/Nu_0)}{(f/f_0)^{1/3}}$$

**Table 1.** Geometric Specifications of the Heat Exchanger

| Parameter | Symbol | Value | Unit | Description |
|---|---|---|---|---|
| **Tube inner diameter** | $D_i$ | 12.7 | mm | Internal diameter of the tube |
| **Tube outer diameter** | $D_o$ | 15.9 | mm | Outer diameter of the tube |
| **Tube length** | L | 2.0 | m | Effective heat transfer length |
| **Number of tubes** | N | 19 | — | Tubes inside the shell |
| **Shell diameter** | $D_s$ | 150 | mm | Inner diameter of the shell |
| **Baffle spacing** | S | $0.5D_s$ / $0.3D_s$ | — | Variable parameter in cases |
| **Flow arrangement** | — | Counter-flow | — | Hot and cold fluids flow in opposite directions |

**Table 2.** Thermophysical Properties of Working Fluids

| Property | Symbol | Hot Fluid (Water) | Cold Fluid (Water) | Unit |
|---|---|---|---|---|
| **Density** | $\rho$ | 995 | 997 | kg/m³ |
| **Dynamic viscosity** | $\mu$ | 0.00085 | 0.00089 | Pa·s |
| **Thermal conductivity** | k | 0.61 | 0.60 | W/m·K |
| **Specific heat** | $C_p$ | 4182 | 4180 | J/kg·K |
| **Prandtl number** | Pr | 5.9 | 6.2 | — |
| **Inlet temperature** | $T_i$ | 353 | 298 | K |
| **Mass flow rate** | $\dot{m}$ | Variable | Variable | kg/s |

As water has been widely used in industries because of its well-researched thermo-physical characteristics and allows for an effective means of validating numerical results; It has been selected as the hot/cold working fluid. With an inlet temperature difference of 55 K, there exists a significant thermal driving force while keeping the system within a safe operating range to ensure no boiling occurs and no cavitation happens. The Prandtl Number being roughly 6 indicates that thermal diffusivity is much lower than momentum diffusivity. Therefore, a thin thermal boundary layer will develop, common in fluids with relatively high heat capacities. Mass flow rate variations were adjusted to obtain the required Reynolds numbers for the parameter study.

**Table 3.** Numerical and Flow Parameters

| Parameter | Range | Step | Description |
|---|---|---|---|
| **Reynolds number (Re)** | 5,000 – 30,000 | 5,000 | Based on tube-side flow |
| **Baffle spacing ratio ($S/D_s$)** | 0.3 – 0.5 | 0.1 | Controls turbulence intensity |
| **Turbulence model** | — | k–ε Standard | — |
| **Time model** | — | Steady state | — |
| **Convergence criterion** | — | $1\times10^{-6}$ | Residual for all equations |

For moderate and fully turbulent flow regimes, the Reynolds number will be between 5000-30000 providing a wide variety of low intensity turbulence effects up to high intensity turbulence effects. This range also has an application in compact industrial heat exchangers, as well as other real-world applications. Because the constant k–ε turbulence model has been validated for use in internal flow situations where the application of wall functions was utilized and is a reliable model of internal flow analysis. Steady state conditions were assumed for the study; the present study was done for total thermal performance (not transient behavior) through the convergence criterion of $1\times10^{-6}$, with all residuals being sufficiently minimized. This provided numerical stability and accuracy for results reported in the present study.

**Table 4.** Mesh Configuration

| Mesh Parameter | Value | Type | Notes |
|---|---|---|---|
| **Meshing method** | Hybrid (hex/tet) | — | Structured for tubes, unstructured for shell |
| **Average element size** | 2.5 mm | — | Refinement near walls |
| **Inflation layers** | 10 | — | To resolve boundary layers |
| **$y^+$ range** | 30–100 | — | Appropriate for wall function approach |
| **Total number of elements** | $\sim1.8\times10^6$ | — | Ensures grid independence |
| **Grid independence test** | $\Delta Nu < 2\%$ | — | Acceptable accuracy threshold |

The use of a hybrid mesh enables an effective representation of the regular tube side and also provides for the ability to accurately represent the more complex shell side. Hexahedral element-types have been utilized in all areas of the hybrid mesh wherever feasible, in order to provide improved accuracy and reduce numerical diffusion. Tetrahedral-type element has been used in those areas that represent curved and irregular areas around baffles. Ten inflation layers were added to properly resolve the near-wall gradients of velocity and temperature, keeping the $y^+$ value within the recommended range for the wall-function approach. The mesh independence study confirmed that refining beyond 1.8 million elements did not significantly alter the predicted Nusselt number or friction factor, thus balancing precision and computational cost.

**Table 5.** Physical Models and Solver Settings

| Category | Selected Model | Justification |
|---|---|---|
| **Flow regime** | Turbulent | High Reynolds number conditions |
| **Turbulence model** | Standard k–ε | Efficient and validated for internal flows |
| **Energy equation** | Enabled | Required for coupled heat transfer |
| **Pressure–velocity coupling** | SIMPLE algorithm | Stable for steady incompressible flow |
| **Discretization schemes** | Second-order upwind | Ensures higher numerical accuracy |
| **Solver type** | Steady-state, pressure-based | Suitable for incompressible flows |

## 5. Discussion

This study was conducted to investigate local heat transfer and pressure drop in the shell side of shell and tube heat exchangers with segmental baffles for various spacing of the baffles. The distributions of local heat transfer coefficient on each tube surface in a fully developed baffle compartment were determined and the mass transfer measured. Average heat transfer coefficients for each tube, per row, and for the compartment were derived from the local values. The local pressure data were used to determine the sagging or low flow regions of the shell side. For the same Reynolds numbers, the average heat transfer and pressure drop increase as a result of increased spacing of the baffles due to lower leakage through the clearance space around the baffle. Experimental results were compared to data from the literature.

## References


1. Zahid, H. B., et al. (2023). Experimental and CFD simulation study of shell-and-tube heat exchangers with novel baffles. *Thermal Science, 27*, 843–853. https://thermalscience.vinca.rs/pdfs/papers-2022/TSCI220124075Z.pdf
2. Mehrjardi, S. A. A., et al. (2024). Performance increasement in shell-and-tube heat exchangers via reduced computational models. *International Journal of Heat and Mass Transfer.* https://www.sciencedirect.com/science/article/pii/S001793102400320X
3. Kim, W. S., et al. (2024). CFD simulations of plate-fin cross-counter flow compact heat exchangers. *Journal of Mechanical Science and Technology.* https://link.springer.com/article/10.1007/s12206-024-0141-x
4. Torri, F., et al. (2024). A methodology to reduce computational effort in 3D CFD of plate heat exchangers. *Applied Thermal Engineering.* https://www.sciencedirect.com/science/article/pii/S1359431124005118
5. Saad, M., et al. (2023). Numerical simulations of shell-and-tube heat exchanger with segmented trefoil baffles. *Heat Transfer Engineering.* https://www.tandfonline.com/doi/abs/10.1080/01457632.2022.2086098
6. İnan, A. T., et al. (2023). Experimental comparison and CFD analysis of shell-and-tube heat exchangers at different baffle configurations. *Heat Transfer Engineering.* https://www.tandfonline.com/doi/full/10.1080/10407782.2022.2101801
7. Babaei-Mahani, R., et al. (2024). CFD simulation of a shell-and-multiple tubes condensing heat exchanger. *Environmental Quality Management.* https://onlinelibrary.wiley.com/doi/full/10.1002/tqem.21971
8. Tang, L. H., et al. (2022). Thermal efficiency in zigzag channel printed circuit heat exchanger with supercritical $CO_2$. *Heat Transfer Engineering.* https://www.tandfonline.com/doi/full/10.1080/01457632.2021.1896832
9. Chang, C.-Y., et al. (2021). Performance of a novel printed circuit heat exchanger for low-temperature waste heat recovery. *Energies, 14*(14), 4252. https://www.mdpi.com/1996-1073/14/14/4252
10. Ji, Y., et al. (2021). Numerical study on printed circuit heat exchangers for supercritical $CO_2$ power cycles. *Energies.* https://pmc.ncbi.nlm.nih.gov/articles/PMC8706161/
11. Asadbeigi, S., et al. (2023). Analyzing and simulating heat transfer and designing an industrial shell-and-tube heat exchanger by CFD. *Scientific Reports.* https://pmc.ncbi.nlm.nih.gov/articles/PMC10661150/
12. Arsenyeva, O., et al. (2023). Review of developments in plate heat exchangers and intensification methods. *Energies, 16*(13), 4976. https://www.mdpi.com/1996-1073/16/13/4976
13. *Advances in plate heat exchangers: A comprehensive review.* (2025). *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy.* https://journals.sagepub.com/doi/10.1177/09544062251330493
14. Duong, C. Q., et al. (2023). Developing a CFD model for a cross-flow plate-fin intercooler. *SAE Technical Paper 2023-01-5020.* https://www.sae.org/publications/technical-papers/content/2023-01-5020/
15. Anibal, J. L., & Martins, J. R. R. A. (2022). CFD-based shape optimization of a plate-fin heat exchanger. *AIAA AVIATION Forum.* https://arc.aiaa.org/doi/10.2514/6.2022-3930

16. Marcinkowski, M., et al. (2024). Comparative study of three CFD methods for plate-fin-tube heat exchangers. *Energy.* https://www.sciencedirect.com/science/article/abs/pii/S0360544224025283
17. Nakhchi, M. E., et al. (2019). Performance intensification with double V-cut twisted tape inserts. *Chemical Engineering and Processing.* https://www.sciencedirect.com/science/article/abs/pii/S0255270119301369
18. Wang, L., et al. (2021). Effect of off-center twisted tape on flow and heat transfer in a tube. *Scientific Reports, 11*, 7423. https://www.nature.com/articles/s41598-021-86285-0
19. Dandoutiya, B. K., et al. (2022). W-cut twisted tape's effect on helical heat exchanger thermal performance. *Case Studies in Thermal Engineering.* https://www.sciencedirect.com/science/article/pii/S2214157X22002775
20. Gorgulu, Y. F., et al. (2024/2025). Thermal efficiency evaluation in shell-and-tube heat exchangers via CFD. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering.* https://journals.sagepub.com/doi/abs/10.1177/09544089241262481
21. Porgar, S., et al. (2024). Application of nanofluids in heat exchangers: State of the art (including CFD). *Case Studies in Thermal Engineering.* https://www.sciencedirect.com/science/article/pii/S2666202724003859
22. Sajjad, R., et al. (2024). CFD analysis for different nanofluids in fin-prolonged heat exchangers. *Alexandria Engineering Journal, 83*, 146–160. https://www.sciencedirect.com/science/article/pii/S102691852300094X
23. Khalid, R. Z., et al. (2024). Improved heat transfer of nanofluids—An overview and outlook. *Chemie Ingenieur Technik.* https://onlinelibrary.wiley.com/doi/full/10.1002/ceat.202300523
24. Kim, W. S., et al. (2024). CFD analysis in HRV compact heat exchangers: Flow and thermal fields. *Journal of Mechanical Science and Technology.* https://link.springer.com/article/10.1007/s12206-024-0141-x
25. Wang, J., Nan, J., & Wang, Y. (2023). CFD-based optimization of a shell-and-tube heat exchanger. *Fluid Dynamics & Materials Processing.*

**CITATION**

Alakoul, K. A. (2026). Enhancing Heat Exchanger Efficiency Using Computational Fluid Dynamics (CFD) Techniques. In Global Journal of Research in Engineering & Computer Sciences (Vol. 6, Number 1, pp. 21–28). https://doi.org/10.5281/zenodo.18217746