



OPEN ACCESS

Volume: 5

Issue: 4

Month: October

Year: 2026

ISSN: 2583-7117

Published: 21.10.2026

Citation:

Modassir Ahmad, Piyush Kumar Verma, "CFD Analysis of the Effects of Fin Geometry and Fluid Velocity on the Performance of a Double-Pipe Heat Exchanger" International Journal of Innovations in Science Engineering and Management, vol. 5, no. 4, 2026, pp. 168-185.

DOI:

10.69968/ijsem.2026v5i4168-185



This work is licensed under a Creative Commons Attribution-Share Alike 4.0 International License

CFD Analysis of the Effects of Fin Geometry and Fluid Velocity on the Performance of a Double-Pipe Heat Exchanger

Modassir Ahmad¹, Piyush Kumar Verma²

¹Research Scholar, Department of Mechanical Engineering, Corporate Institute of Science & Technology, Bhopal

²Assistant Professor, Department of Mechanical Engineering, Corporate Institute of Science & Technology, Bhopal

Abstract

Double-pipe heat exchangers are widely used in thermal systems, where improving heat transfer while controlling pressure losses is an important design consideration. The present study investigates the thermal and hydraulic performance of a double-pipe heat exchanger equipped with twisted-fin inserts. The objective was to examine the influence of fin geometry and hot- and cold-fluid velocity variation on heat-transfer performance. Three-dimensional CFD simulations were performed using CATIA V5 for geometry development and ANSYS Fluent 2023 for numerical analysis, with distilled water as the working fluid. The performance was evaluated using outlet temperature, pressure drop, friction factor, heat-transfer rate, convective heat-transfer coefficient, and Nusselt number. Among the investigated fin geometries, Case 4 provided the most suitable geometry, while Case 4.1 produced the highest heat-transfer rate, convective heat-transfer coefficient, and Nusselt number, with values of 2191.85 W, 2207.95, and 84.02, respectively. Case 4.5 provided the highest outlet temperatures. The study demonstrates that twisted-fin geometry and flow conditions strongly influence heat-transfer performance. Future work can investigate nanofluids, alternative tube materials, and other fin geometries to further enhance performance.

Keywords: Double-pipe heat exchanger, Twisted-fin inserts, Heat transfer enhancement, Computational fluid dynamics (CFD), Nusselt number

INTRODUCTION

Heat exchangers are important thermal engineering devices used for transferring heat between two fluids at different temperatures without allowing them to mix directly. Their efficient operation is essential in many engineering and industrial applications, including power generation, refrigeration and air-conditioning, chemical processing, petroleum industries, automotive thermal management, renewable energy systems, and process industries, where effective utilization of thermal energy directly influences energy consumption and system performance. Among the different types of heat exchangers, the double-pipe heat exchanger is one of the comparatively simple and flexible configurations, consisting of two concentric passages through which hot and cold fluids flow. Its relatively simple construction, ease of operation, and suitability for experimental and numerical investigations make it an important configuration for studying heat-transfer behaviour. However, the heat-transfer performance of a conventional double-pipe heat exchanger is influenced by several factors, including fluid properties, temperature difference, flow rate, velocity, flow regime, heat-transfer surface area, and the characteristics of the heat-transfer surface. Therefore, improving the heat-transfer rate without causing an excessive increase in pressure drop remains an important consideration in heat-exchanger design. (Rifqi & Al-Khalil, 2025) (Kotian et al., 2020)(Bhattacharjee, 2020)(Navindgi, 2018)

One of the commonly adopted passive techniques for enhancing heat transfer is the use of extended surfaces or fins. Fins increase the effective heat-transfer area and can also modify the fluid-flow pattern near the heat-transfer surface, thereby influencing the development of thermal boundary layers and the rate of convective heat transfer. The geometry and arrangement of fins play an important role in determining the thermal and hydraulic behaviour of a heat exchanger. Different fin geometries can produce different levels of surface area, flow disturbance, mixing, and turbulence, resulting in variations in heat-transfer performance. At the same time, fluid velocity has a direct influence on convective heat transfer because changes in velocity alter the Reynolds number, flow regime, boundary-layer development, and interaction between the flowing fluid and the heated surface. Increasing velocity may enhance heat transfer, but it may also increase pressure loss and pumping requirements. (Omidi et al., 2017)(Kadhim et al., 2025)(Arjmandi et al., 2020) Thus, the combined influence of fin geometry and fluid velocity needs to be considered when evaluating the performance of a finned double-pipe heat exchanger. The present study therefore focuses on examining the effect of different fin geometries and fluid velocities on the heat-transfer performance of a double-pipe heat exchanger, with emphasis on understanding the resulting thermal and flow characteristics and identifying conditions that provide improved heat-transfer performance while maintaining acceptable hydraulic behaviour.

Double Pipe Heat Exchanger (DPHE) and Working Principle

A double-pipe heat exchanger (DPHE), also known as a concentric-tube, pipe-in-pipe, or hairpin heat exchanger, consists of two concentric pipes of different diameters, through which two fluids flow separately. One fluid passes through the inner tube, while the other flows through the annular space between the inner and outer pipes. The wall of the inner tube serves as the primary heat-transfer surface, allowing heat to transfer from the hot fluid to the cold fluid through a combination of convection and conduction. The exchanger may be operated in either parallel-flow or counterflow arrangement. In parallel flow, both fluids move in the same direction, whereas in counterflow, they move in opposite directions. Counterflow operation generally provides a more favorable temperature gradient along the exchanger and can achieve higher thermal effectiveness. The basic construction includes an inner tube, outer pipe, fluid inlet and outlet connections, supports, and, where required, insulation to minimize heat loss. Depending on the operating

conditions, materials such as stainless steel, copper, and carbon steel can be selected based on thermal conductivity, mechanical strength, temperature, pressure, and fluid compatibility. DPHEs are relatively simple to manufacture, install, maintain, and modify, making them suitable for various heating and cooling applications. Their thermal performance can further be enhanced by incorporating fins or other flow-modifying elements on the heat-transfer surface, which increase the effective area and promote fluid mixing, thereby improving heat transfer. (Gabir & Alkhafaji, 2021)(Kumbhar et al., 2023)(Chaudhari & Adroja, 2014)(Singh et al., 2022) (Usmani et al., 2025)

Role of Extended Surfaces and Fins in Heat Transfer Enhancement

A fin is an extended surface attached to a solid body to increase the rate of heat transfer between the surface and the surrounding fluid. The basic principle behind fin application is to provide additional surface area through which heat can be transferred by convection. The amount of heat transferred from a fin depends on factors such as the temperature difference between the fin and the surrounding fluid, the convective heat-transfer coefficient, the thermal conductivity of the fin material, and its available surface area. When increasing the temperature difference or modifying the fluid-flow conditions is impractical or uneconomical, extending the surface area through fins provides an effective passive approach for improving heat dissipation. (Ali et al., 2023)(Aridi et al., 2022)(Talakonda et al., 2019)(El Hassan et al., 2025) (Haw et al., 2025)

In double-pipe heat exchangers, extended surfaces can be incorporated on the inner tube or within the annular passage to provide additional contact area between the heat-transfer surface and the working fluid. This approach can reduce the effective thermal resistance and improve the overall heat-transfer rate without requiring a substantial increase in the heat-exchanger dimensions. Various fin arrangements, including longitudinal, annular, helical, rectangular, pin and perforated configurations, have been investigated because their geometry can influence fluid mixing and thermal boundary-layer development. However, fin enhancement also modifies the flow passage and may increase flow resistance. Excessive fin height, thickness or density can therefore result in higher frictional losses and pressure drop, increasing pumping-power requirements. Consequently, effective fin design requires appropriate selection of geometry, spacing, material and arrangement to achieve improved heat transfer while maintaining acceptable

hydraulic performance. (Dandoutiya & Kumar, 2023)(Rostami & Ahmadi, 2025)(Lavakumar et al., 2024)

Effect of Reynolds Number and Flow Regime on Heat Transfer Performance

The Reynolds number is a dimensionless parameter that plays an important role in the design and analysis of fluid flow and heat transfer systems because it helps characterize the prevailing flow regime. Its value influences both hydraulic and thermal performance. As the Reynolds number increases, the flow may transition from laminar to turbulent conditions, resulting in greater fluid mixing and enhanced heat transfer. However, turbulent flow also produces higher frictional losses and consequently a greater pressure drop, which increases the pumping power required to maintain the desired flow rate. The Reynolds number is also important in the selection and accurate operation of flow-measuring devices, particularly differential-pressure flowmeters such as orifice plates, because their performance depends on the flow regime and the development of the flow profile. In addition, Reynolds number similarity is essential when developing and testing scaled physical models. Maintaining comparable Reynolds numbers between a model and its corresponding full-scale system helps reproduce similar flow characteristics and provides a reliable basis for predicting fluid-flow and heat-transfer behavior in practical applications. (Savas, Evran & Kurt, 2023) (Hameed & Awad, 2021) (Rai & Muchrikar, 2021) (Kotian et al., 2020)

RESEARCH OBJECTIVE

1. To design and model a conventional double pipe heat exchanger and modified configurations with different fin shapes attached to the pipe surface.
2. To analyze the influence of fin geometry variations, such as shape and arrangement, on the heat transfer performance of the double pipe heat exchanger.
3. To study the effect of varying inlet velocities of hot and cold fluids on outlet temperatures and overall heat transfer characteristics.
4. To evaluate the outlet temperatures of both hot and cold fluids for different geometric and flow conditions.

5. To determine the Nusselt number and convective heat transfer coefficient for each fin configuration.
6. To analyze the friction factor and pressure drop associated with different fin geometries and flow velocities.
7. To calculate and compare the rate of heat transfer for the modified and conventional heat exchanger models.
8. To identify the optimal fin geometry and operating conditions that provide maximum heat transfer enhancement with acceptable frictional losses.

RESEARCH METHODOLOGY

Steps of Working

The present numerical investigation follows a systematic methodology beginning with a detailed review of relevant research studies to identify the research gap and select a suitable reference model. The geometry of the selected model is developed using CATIA V5 and imported into ANSYS Fluent for computational analysis. The computational domain is then generated and discretized using an appropriate mesh, followed by the specification of material properties, numerical models, and boundary conditions based on the reference study. After completing the simulation, the required thermal and hydraulic parameters are extracted and compared with the reported reference results to verify the reliability of the developed model. If significant differences are observed, the relevant modelling parameters, boundary conditions, or numerical settings are reviewed and the simulation is repeated until satisfactory agreement is obtained. Once the model is validated, the identified research gap is incorporated according to the objectives of the present study. Different configurations are subsequently developed by systematically varying the selected geometrical or operating parameters. Each case is simulated under identical prescribed conditions, and the obtained results are analyzed comparatively to evaluate their effects on heat transfer and hydraulic performance and to identify the configuration with the desired performance characteristics.

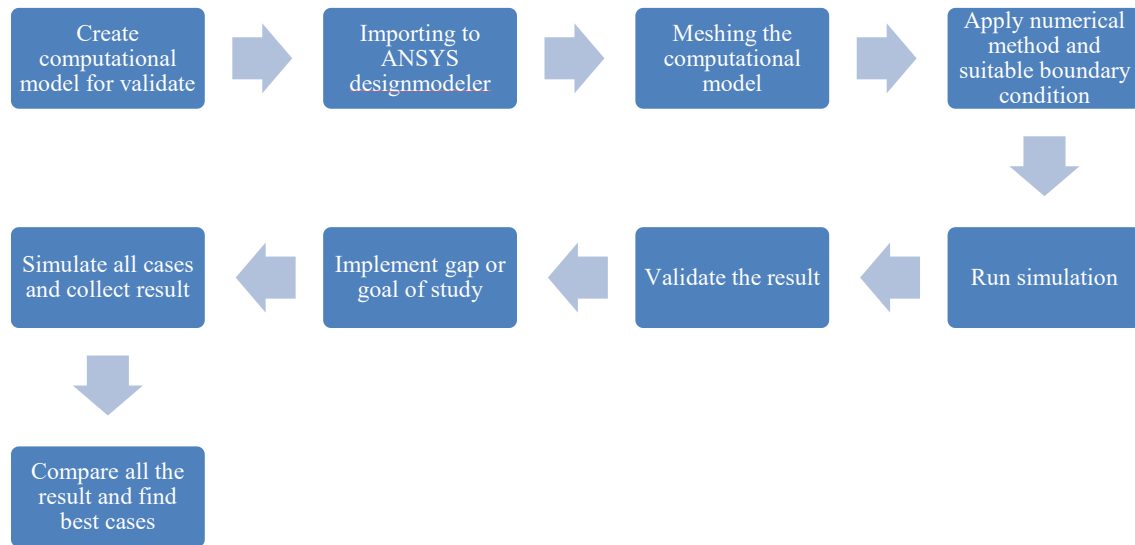


Figure 1: Flow diagram of study

Geometry Modeling

The computational model comprises a double-pipe heat exchanger (DPHE) with different twisted-fin (TF) configurations incorporated on the outer surface of the inner tube to investigate their effects on thermal and hydraulic performance. The DPHE has a total length of 1700 mm and consists of concentric inner and outer pipes separated by a steel wall. The hot fluid flows through the inner tube, whereas the cold fluid passes through the annular region. The hydraulic diameters of the cold-fluid domain, hot-fluid domain, and separating steel wall are 23.1 mm, 10 mm, and 2.75 mm, respectively, as shown in Figure 2. A 100 mm buffer region is provided at both the inlet and outlet to facilitate flow development, while the central analysis zone has a length of 1500 mm. Five different fin geometries are developed and incorporated within this region, and all configurations are analyzed under identical operating and numerical conditions for systematic comparison.

The first configuration consists of six sets of rectangular twisted fins, with four equally distributed fins in each set. The first set is positioned 250 mm from the beginning of the analysis zone, followed by a 200 mm pitch. Each fin has a width of 1.2 mm, length of 10 mm, height of 4 mm, and twist

angle of 40°. The second configuration uses single helical fins with a 50 mm pitch, one revolution, and 25 mm height, arranged counterclockwise. Nine fin sets are positioned at 150 mm intervals, with the first set located 150 mm from the inner-tube inlet. The third configuration employs two equally distributed helical fins with a 70 mm pitch and 35 mm height, while the fourth uses the same dual-fin arrangement with a 50 mm pitch and 25 mm height. The fifth configuration also incorporates dual helical fins with a 50 mm pitch, one revolution, and 25 mm height, arranged in nine equally spaced sets.

Table 1: Geometric parameters of the DPHE and TF in the numerical study. (Haw et al., 2025)

Parameters	Values
Total length	1700 mm
Analysis length	1500 mm
Cold fluid domain outer diameter	35.85 mm
Cold fluid domain inner diameter	12.75 mm
Steel wall outer diameter	12.75 mm
Steel wall inner diameter	10 mm
Hot domain diameter	10 mm

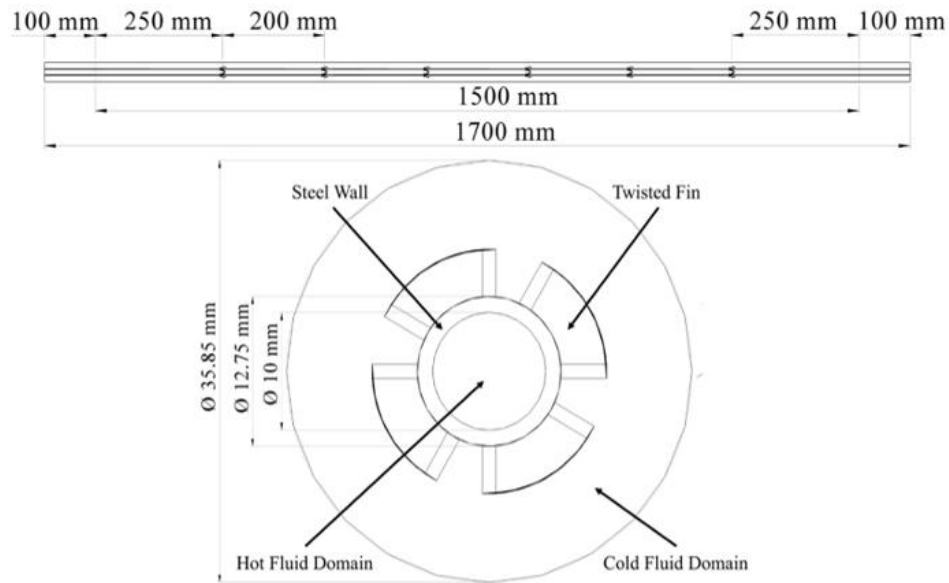


Figure 2: CAD model of the DPHE and TF geometry, illustrating the structural configuration and dimensions
(Haw et al., 2025)

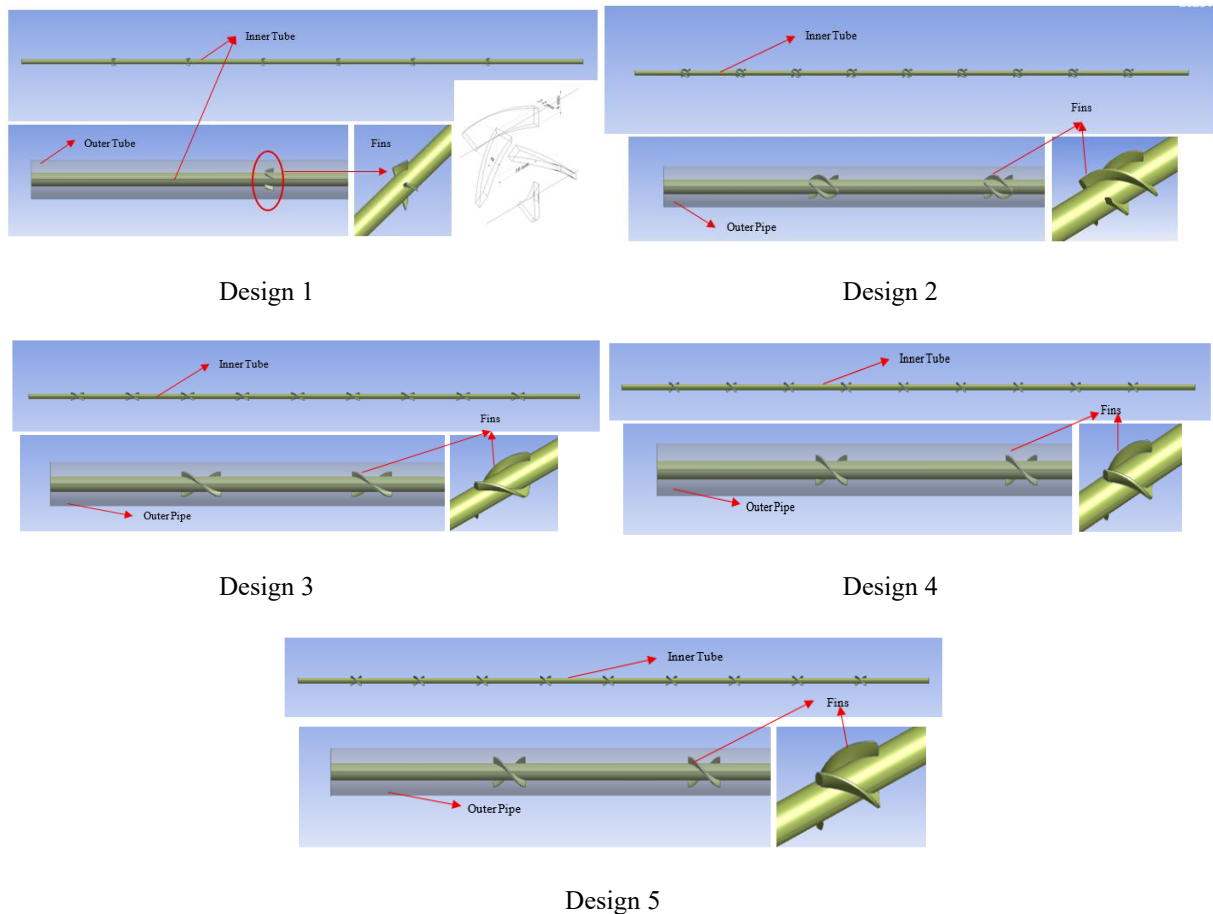


Figure 3: Geometry configuration of Design 1-5

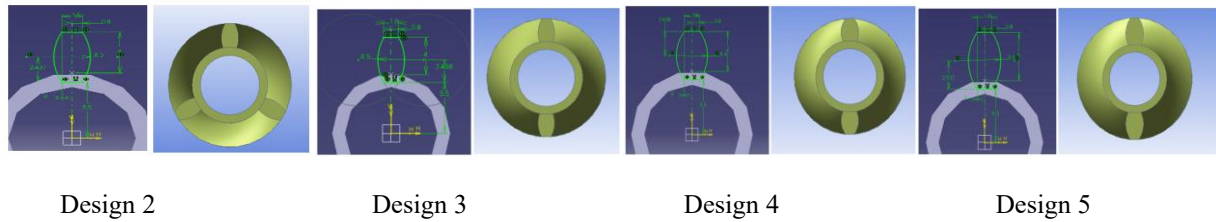


Figure 4: Fin Geometry of design 2-5

Meshing generation

The computational models of the double-pipe heat exchanger with different twisted-fin configurations were discretized into finite computational elements using ANSYS Meshing. Tetrahedral and hexahedral elements were employed to represent the fluid and solid domains and to accurately capture the complex geometry of the twisted fins. The mesh was refined around the twisted-fin surfaces and fluid–solid interfaces, where significant changes in velocity and temperature are expected. Adequate mesh resolution in these regions is important for accurately predicting the heat-transfer and flow characteristics. Due to the variation in

geometry among the five twisted-fin configurations, the number of elements and nodes differed for each computational model. The total number of elements ranged from 645,637 to 843,978, while the number of nodes varied from 193,836 to 233,201. The difference in mesh size was mainly associated with the geometric complexity and arrangement of the twisted fins. The generated meshes provided sufficient spatial discretization of the computational domains for the numerical evaluation of the thermal and hydraulic performance of the different configurations. Figure 5 illustrate the meshing of computational model of double pipe heat exchanger.

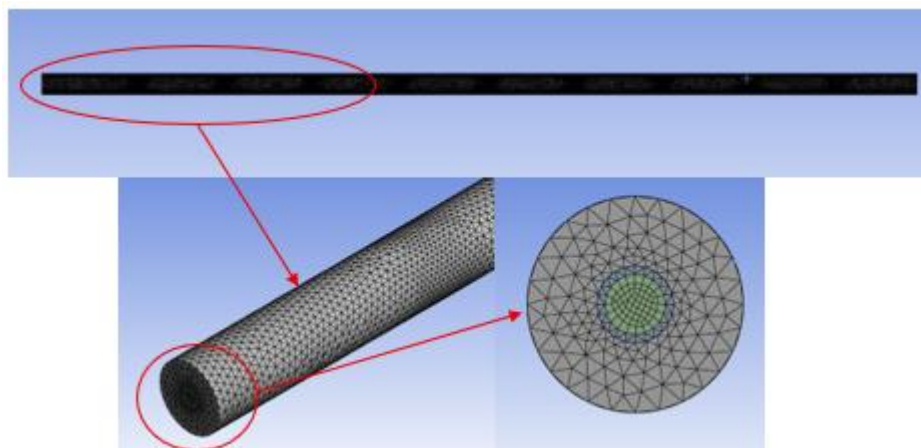


Figure 5: Meshing of computational model

Numerical Simulations and Boundary Condition

The numerical investigation was performed using ANSYS Fluent 2023 to evaluate the thermal-hydraulic performance of twisted-fin (TF) inserts in a double-pipe heat exchanger (DPHE) using distilled water as the heat-transfer fluid. The cold and hot fluid inlet temperatures were maintained at 25°C and 50°C, respectively. Mass-flow-rate boundary conditions were applied to both fluid domains, with the cold-fluid mass flow rate varied from 0.14 to 0.234 kg/s and the hot-fluid mass flow rate from 0.13 to 0.165 kg/s. The temperature-dependent thermophysical properties of distilled water, including density, viscosity, specific heat,

and thermal conductivity, were obtained from Engineering Toolbox, while the default steel properties available in ANSYS Fluent 2023 were used for the separating wall. The analysis primarily focused on the turbulent flow behavior and thermal performance of the cold-fluid domain containing the TF inserts.

A mass-flow inlet and pressure outlet were specified at the respective inlet and outlet boundaries of both fluid domains, while no-slip conditions were imposed on all solid surfaces. The simulations were conducted under steady-state, three-dimensional, incompressible-flow conditions, with gravity effects included. Turbulence was modeled

using the SST $k-\omega$ model to effectively capture the flow separation, mixing, and complex turbulent structures generated by the TF inserts. The COUPLED pressure-velocity solver with second-order upwind spatial discretization was employed to obtain stable and accurate solutions of the governing equations. Pseudo-transient relaxation was used to improve convergence stability, and residual convergence criteria were set to 10^{-6} , with a minimum residual threshold of 10^{-3} for the monitored equations. A summary of the applied simulation settings and boundary conditions is presented in Table 2.

Table 2: Numerical simulation settings and applied boundary conditions. (Haw et al., 2025)

Variables	Properties
Viscous Model	SST $k-\omega$
Gravity	9.81 m/s ²
CFD Algorithm	COUPLED
Interpolating Scheme	Second Order Upwind
Cold Fluid Domain	
Inlet type	Mass Flow rate
Inlet Temperature	25°C
Inlet mass flow rate	0.14 – 0.234 kg/s
Fluid Exit	Pressure Outlet
Outer Wall	Adiabatic
Inner Wall	Conductive Wall
Fin Wall	Conductive Wall
Hot Fluid Domain	
Inlet type	Mass Flow rate
Inlet Temperature	50°C
Inlet mass flow rate	0.13 – 0.165 kg/s

Continuity equation:

$$\frac{1}{r} \frac{\partial(rV_r)}{\partial r} + \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} = 0$$

Momentum equation with respect to r:

$$\frac{\partial V_r}{\partial t} + V_r \frac{\partial V_r}{\partial r} + \frac{V_\theta}{r} \frac{\partial V_r}{\partial \theta} + V_z \frac{\partial V_r}{\partial z} - \frac{V_\theta^2}{r} = -\frac{1}{\rho} \frac{\partial p}{\partial r} + \frac{\mu}{\rho} \left[\frac{\partial}{\partial r} \left(\frac{1}{r} \frac{\partial}{\partial r} (rv_r) \right) + \frac{1}{r^2} \frac{\partial^2 V_r}{\partial \theta^2} + \frac{\partial^2 V_r}{\partial z^2} - \frac{2}{r^2} \frac{\partial V_r}{\partial \theta} \right] + S_r$$

Momentum equation with respect to θ :

$$\frac{\partial V_\theta}{\partial t} + V_r \frac{\partial V_\theta}{\partial r} + \frac{V_\theta}{r} \frac{\partial V_\theta}{\partial \theta} + V_z \frac{\partial V_\theta}{\partial z} + \frac{V_r V_\theta}{r} = -\frac{1}{\rho r} \frac{\partial p}{\partial \theta} + \frac{\mu}{\rho} \left[\frac{\partial}{\partial r} \left(\frac{1}{r} \frac{\partial}{\partial r} (rv_\theta) \right) + \frac{1}{r^2} \frac{\partial^2 V_\theta}{\partial \theta^2} + \frac{\partial^2 V_\theta}{\partial z^2} - \frac{2}{r^2} \frac{\partial V_r}{\partial \theta} \right] + S_\theta$$

Momentum equation with respect to z:

$$\frac{\partial V_z}{\partial t} + V_r \frac{\partial V_z}{\partial r} + \frac{V_\theta}{r} \frac{\partial V_z}{\partial \theta} + V_z \frac{\partial V_z}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \frac{\mu}{\rho} \left[\frac{\partial}{\partial r} \left(r \frac{\partial v_z}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 V_z}{\partial \theta^2} + \frac{\partial^2 V_z}{\partial z^2} \right] + S_z$$

Fluid Exit	Pressure Outlet
Wall	Conductive Wall

Governing Equation

The governing Navier-Stokes equations for mass, momentum, and energy conservation are formulated in cylindrical coordinates to align with the simulation model's setup, as described in Equations (1) – (6). The COUPLED scheme solver addressed the three-dimensional Navier-stokes equations under several simplifying assumptions.

- The fluid flow is considered steady-state and incompressible, with transient effects and density variations due to pressure changes are neglected.
- The thermophysical properties of the distilled water are assumed to be constant. Variations in properties due to temperature fluctuations are not considered.
- Gravity effects are included to account for buoyancy forces in the fluid domain.
- A no-slip boundary condition for the walls is assumed, stating that the fluid adheres completely to all solid surfaces, resulting in zero relative velocity between the fluid and the heat exchanger walls.
- Fouling and surface roughness are neglected, and the wall surfaces are assumed to be hydraulically smooth, stating that the effects of fouling and surface roughness on flow and heat transfer performance are not considered.

Energy equation:

$$\frac{\partial i}{\partial t} + \frac{1}{r} \frac{\partial(rV_r i)}{\partial r} + \frac{1}{r} \frac{\partial(V_\theta i)}{\partial r} + \frac{\partial(V_z i)}{\partial z} = -\frac{1}{\rho} \rho \nabla \cdot u + \frac{1}{\rho} \nabla \cdot (k \nabla T) + \frac{1}{\rho} \Phi + \frac{1}{\rho} S_i$$

The internal energy of the fluid is represented by the i term, whereas the S term denotes the sources term determined by the initial conditions. The energy dissipation function Φ is defined in Equation (6).

$$\Phi = \mu \left\{ 2 \left[\left(\frac{\partial V_r}{\partial r \cos \theta} \right)^2 + \left(\frac{\partial V_\theta}{\partial r \sin \theta} \right)^2 + \left(\frac{\partial V_z}{\partial z} \right)^2 \right] + \left(\frac{\partial V_r}{\partial r \sin \theta} + \frac{\partial V_\theta}{\partial r \cos \theta} \right)^2 + \left(\frac{\partial V_r}{\partial z} + \frac{\partial V_z}{\partial r \cos \theta} \right)^2 + \left(\frac{\partial V_\theta}{\partial z} + \frac{\partial V_z}{\partial r \sin \theta} \right)^2 \right\} + \lambda (\nabla \cdot u)^2$$

Case Description

The numerical analysis of heat transfer in the double-pipe heat exchanger (DPHE) was conducted in two major stages, comprising a total of 11 configurations. The first stage focused on the effect of twisted-fin geometry, in which five different fin configurations were investigated and designated as Case 1 to Case 4. These cases were developed by varying the geometry and arrangement of the twisted fins while maintaining the other operating conditions constant. The second stage examined the effect of flow velocity variation in both the hot- and cold-fluid domains to evaluate its influence on the thermal and hydraulic performance of the DPHE. The details of all the investigated configurations, including their respective fin arrangements and operating conditions, are presented in the table 3.

Table 3: Case description

Case	Design Configuration	Mass Flow Rate (kg/s)	
		Hot fluid	Cold fluid
Validation	Desing 1	0.165	0.234
Case 1	Desing 2		
Case 2	Desing 3		
Case 3	Desing 4		
Case 4	Desing 5	0.165	0.234
Case 4.1		0.2	0.234
Case 4.2		0.2	0.18
Case 4.3		0.165	0.18
Case 4.4		0.165	0.14
Case 4.5		0.2	0.14
Case 4.6		0.13	0.14

Validation

Validation is an essential stage of the numerical investigation because it establishes the reliability and accuracy of the developed computational model before proceeding to the performance enhancement analysis. In the present study, the double-pipe heat exchanger (DPHE) geometry described in Section 3.2 (Geometry Modeling) was selected for validation. The reference configuration, designated as Design 1, was developed based on the

geometry reported by Haw et al. (2025) (Haw et al., 2025). The computational geometry was developed using CATIA V5, while the numerical simulations were performed using ANSYS Fluent. The numerical and boundary conditions applied during the simulation were consistent with those described in Section 3.4 (Numerical and Boundary Conditions). The friction factor was selected as the primary parameter for validating the computational model. The friction-factor results obtained from the present CFD simulation were compared with the corresponding results reported by Haw et al. (2025) (Haw et al., 2025). This comparison was performed to verify the accuracy of the numerical methodology and the developed CFD model in predicting the flow characteristics of the DPHE with twisted fins. As illustrated in Figure 6, the present simulation results show satisfactory agreement with the results reported by Haw et al. (2025) (Haw et al., 2025), indicating that the developed computational model provides an acceptable representation of the reference heat-exchanger configuration. Therefore, the validated model was considered suitable for subsequent numerical investigations aimed at evaluating and enhancing the thermal-hydraulic performance of the DPHE.

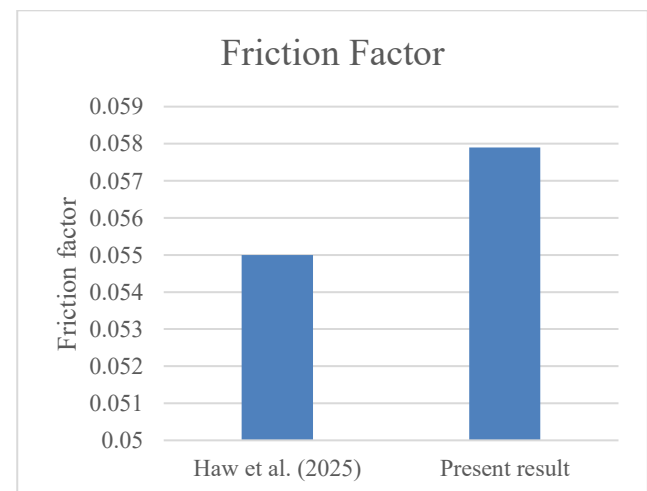


Figure 6: validation of double pipe heat exchanger

RESULT AND DISCUSSION

This chapter presents the numerical results obtained from the CFD investigation of a double-pipe heat exchanger (DPHE) equipped with different twisted-fin configurations. The study evaluates the effects of fin geometry and flow conditions on thermal and hydraulic performance using outlet temperature, pressure drop, friction factor, heat-transfer rate, convective heat-transfer coefficient, and Nusselt number. The investigation was conducted in two parts. The first part examined the effect of fin geometry using four configurations (Cases 1–4) under identical hot- and cold-fluid mass flow rates. Based on the comparative thermal and hydraulic performance, one configuration was selected for further analysis. The second part investigated the effect of hot- and cold-fluid flow-rate variations using the selected configuration through six additional cases (Cases 4.1–4.6). The resulting performance variations are discussed systematically in the following sections.

Effect of Fin Geometry

Table 4: Outlet temperature of cold and hot fluid in case 1–4

Parameter	Case 1	Case 2	Case 3	Case 4
Cold O/L Temp.	300.2	300.23	300.304	300.349
Hot O/L Temp.	320.325	320.262	320.145	320.106
Pressure Drop Cold	216.869	174.247	203.655	254.232
Pressure Drop Hot	8113.15	8113.05	8113.13	8113.1
Friction factor (F)	0.093	0.0748	0.0875	0.109
Heat Transfer rate (Q) Cold	2046.2	2035.59	2107.2	2151.84
Heat Transfer rate (Q) Hot	1989.28	1992.58	2072.92	2100.27
Heat Transfer rate (Q)	2017.74	2014.09	2090.06	2126.06
Convective Heat transfer	2011.27	2014.17	2154.27	2206.88
Nusselt Number (Nu)	76.541	76.6513	81.9828	83.985

The effect of twisted-fin geometry on the thermal and hydraulic performance of the double-pipe heat exchanger

(DPHE) was investigated using Cases 1–4, corresponding to Designs 2–5, respectively. The hot- and cold-fluid mass flow rates were maintained at 0.165 kg/s and 0.234 kg/s, respectively, while the geometry and arrangement of the twisted fins were varied. This comparison was performed to determine how changes in fin configuration influence fluid mixing, heat-transfer characteristics and flow resistance. The results show that the fin geometry has a noticeable influence on outlet temperature, pressure drop, friction factor, heat-transfer rate, convective heat-transfer coefficient and Nusselt number. Among the investigated configurations, Case 4 produced the highest overall heat-transfer parameters, whereas the hydraulic resistance also increased because of the stronger flow disturbance produced by its fin arrangement.

Outlet Temperature

The outlet-temperature results for Cases 1–4 show a gradual change in the thermal response of both fluids with variation in fin geometry. The cold-fluid outlet temperature increased from 300.200 K in Case 1 to 300.349 K in Case 4, while the hot-fluid outlet temperature decreased from 320.325 K to 320.106 K over the same cases. Thus, Case 4 produced the highest cold-fluid outlet temperature and the lowest hot-fluid outlet temperature, indicating comparatively greater thermal energy transfer from the hot fluid to the cold fluid. The observed behaviour can be attributed to the influence of fin geometry on fluid mixing and thermal-boundary-layer development. A suitable twisted-fin arrangement increases disturbance in the flow and promotes interaction between the fluid and heat-transfer surface. This disturbance reduces the persistence of the thermal boundary layer and increases the effective convective heat-transfer process. Consequently, a greater quantity of heat can be transferred from the hot stream to the cold stream. The relatively small difference in outlet temperatures between the cases indicates that the change in fin geometry modifies the heat-transfer process progressively rather than producing a drastic change in the bulk fluid temperature.

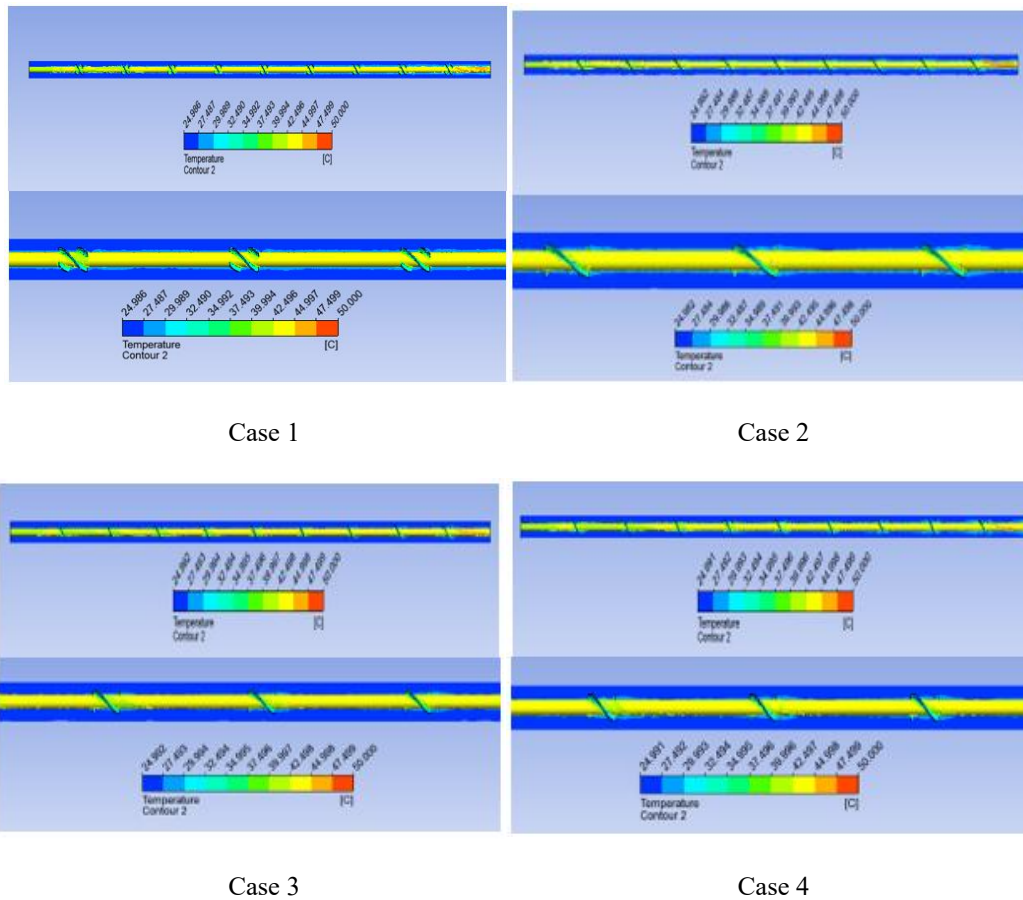


Figure 7: Temperature contour of case 1-4

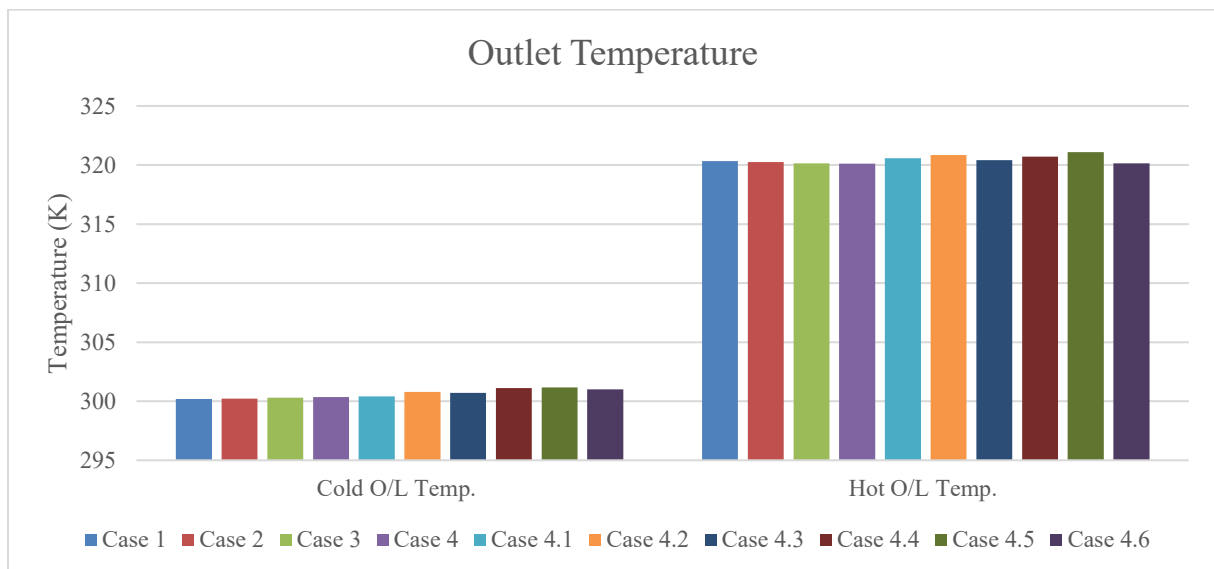


Figure 8: Outlet temperature of cold and hot fluid in case 1-4 and case 4.1-4.6

Pressure Drop and Friction Factor

The hydraulic results demonstrate a stronger influence of fin geometry on flow resistance. The cold-fluid pressure

drop was 216.869 Pa in Case 1, decreased to 174.247 Pa in Case 2, increased to 203.655 Pa in Case 3, and reached 254.232 Pa in Case 4. The corresponding friction factors

were 0.093, 0.0748, 0.0875 and 0.109, respectively. In contrast, the hot-fluid pressure drop remained almost constant at approximately 8113 Pa for all four configurations.

The variation in cold-side pressure drop and friction factor is associated with differences in the geometry and arrangement of the twisted fins. Fin-induced flow obstruction changes the available flow area and generates secondary motion, flow separation and additional mixing.

These effects increase momentum losses within the flow passage. Case 4 exhibited the highest friction factor and cold-side pressure drop, indicating that its fin configuration generated the greatest hydraulic resistance among the investigated geometries. Although this resistance represents an additional pumping-power requirement, the increased flow disturbance also contributed to improved heat transfer. Therefore, the hydraulic penalty observed in Case 4 should be considered together with its thermal enhancement rather than independently.

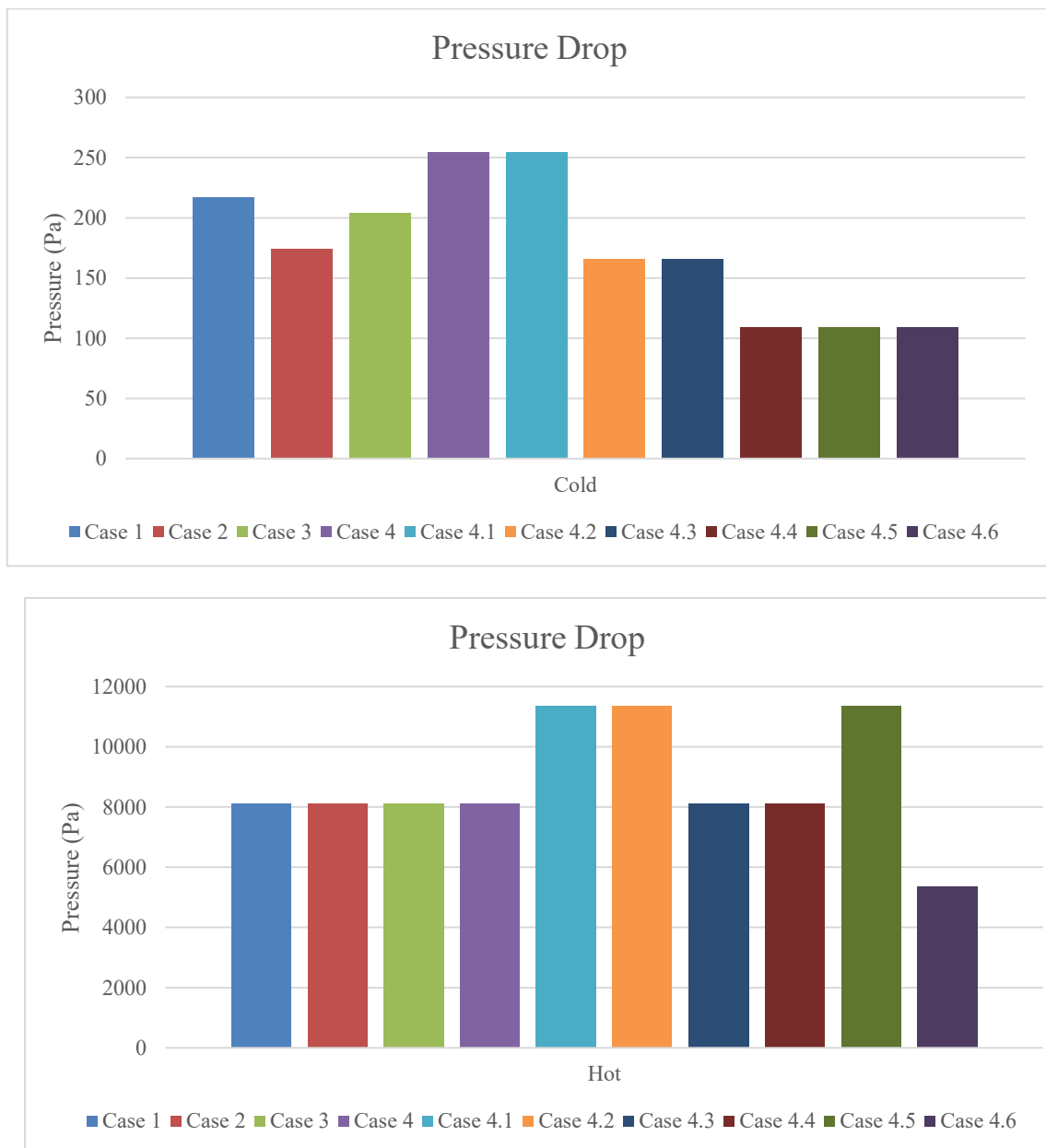


Figure 9: Pressure drop of cold fluid and hot fluid in case 1-4 and case 4.1-4.6

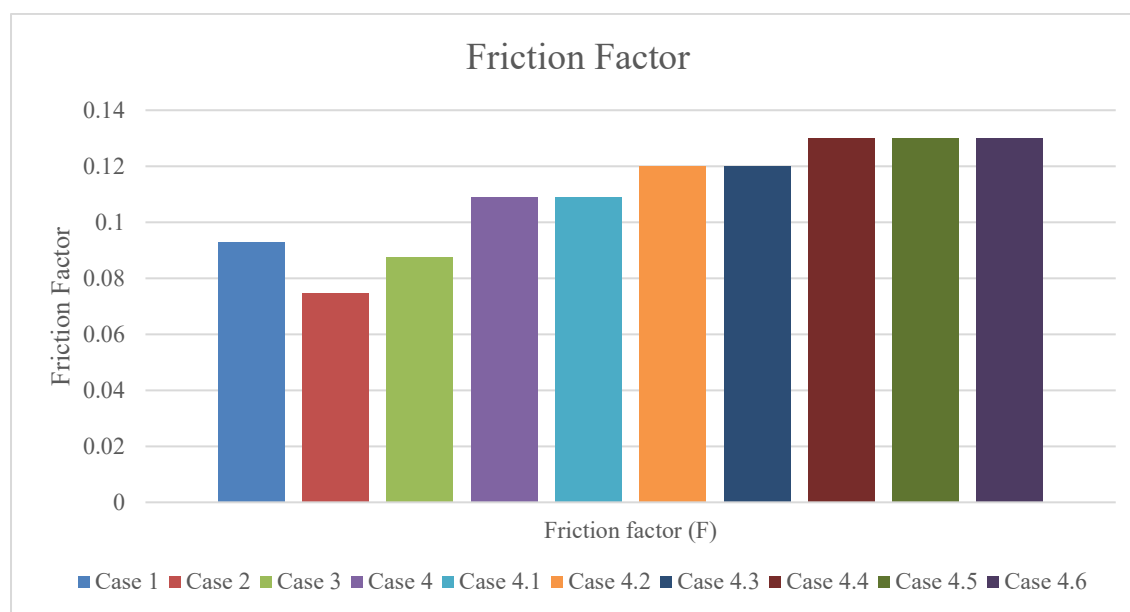


Figure 10: Friction factor in case 1-4 and case 4.1-4.6

Heat Transfer Rate, Convective Heat Transfer Coefficient and Nusselt Number

The thermal results indicate a progressive improvement in heat-transfer performance with the change in fin configuration, particularly for Case 4. The average heat-transfer rate increased from 2017.74 W in Case 1 to 2014.09 W in Case 2, 2090.06 W in Case 3, and 2126.06 W in Case 4. The convective heat-transfer coefficient increased from 2011.27 W/m²K in Case 1 to 2206.88 W/m²K in Case 4, while the Nusselt number increased from 76.541 to 83.985. Case 4 therefore provided the highest values of the principal thermal-performance parameters among the four configurations.

The enhancement can be explained by the ability of the twisted-fin geometry to disturb the developing thermal

boundary layer and promote secondary flow. Such flow disturbances increase fluid mixing and improve the interaction between the fluid and heat-transfer surface. The resulting reduction in thermal resistance increases the convective heat-transfer coefficient and consequently the Nusselt number. The heat-transfer-rate results also show that the improved thermal characteristics are reflected in the overall heat transferred by the exchanger. Although Case 2 shows a slightly lower heat-transfer rate than Case 1, the subsequent increase observed in Cases 3 and 4 indicates that the particular geometry and arrangement of the fins strongly influence the intensity of flow disturbance. Case 4, with the highest Nusselt number and convective heat-transfer coefficient, was therefore selected for the subsequent investigation of fluid-velocity effects.

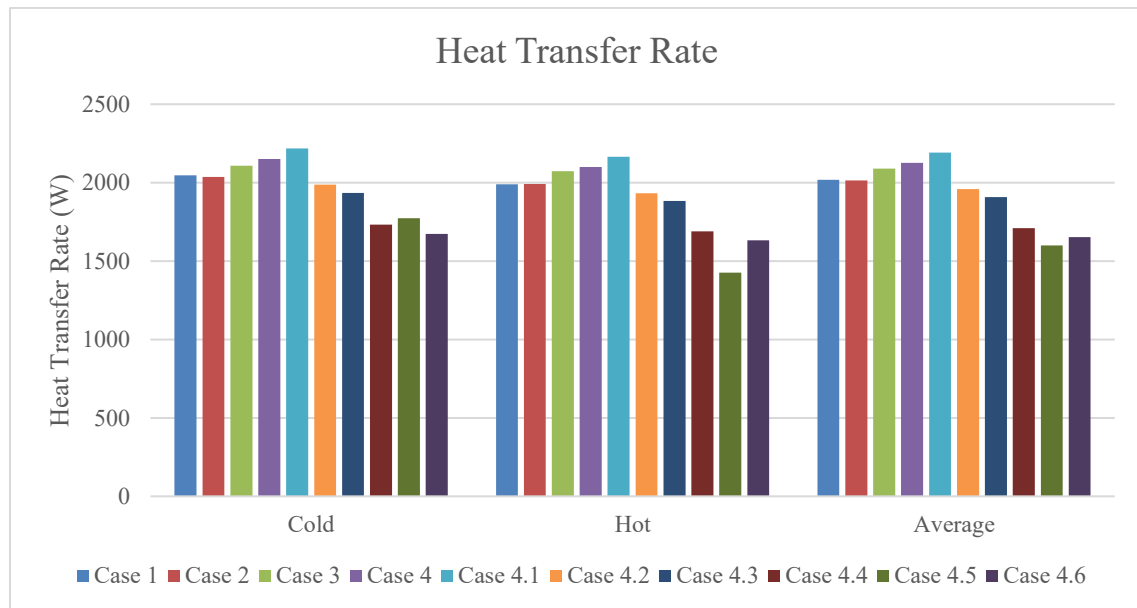


Figure 11: Heat Transfer Rate of cold fluid, hot fluid and Average in case 1-4 and case 4.1-4.6

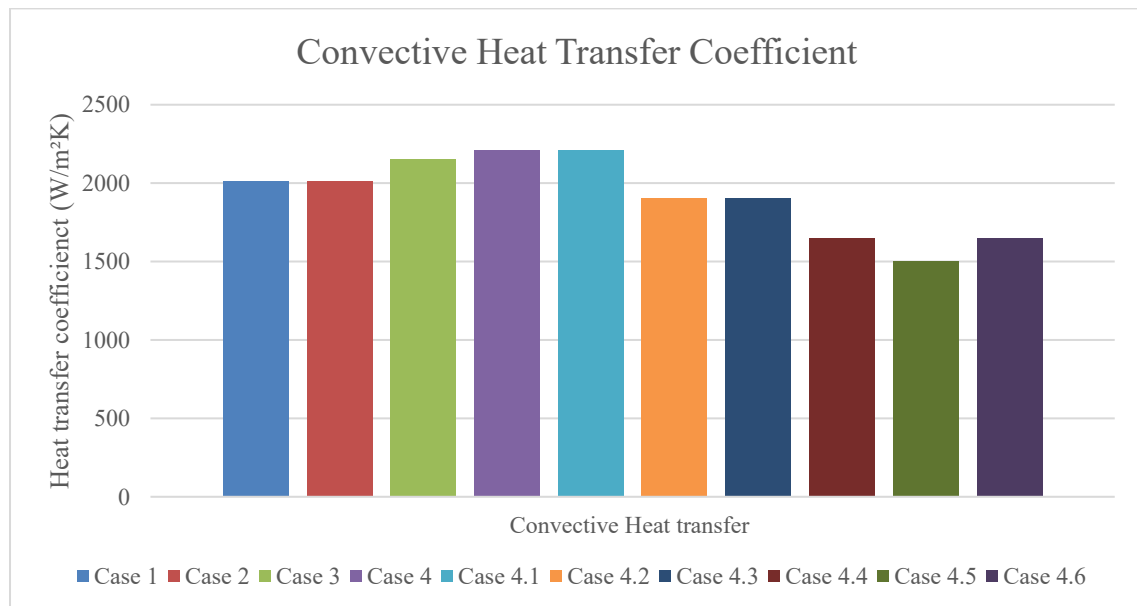


Figure 12: Convective heat transfer coefficient in case 1-4 and case 4.1-4.6

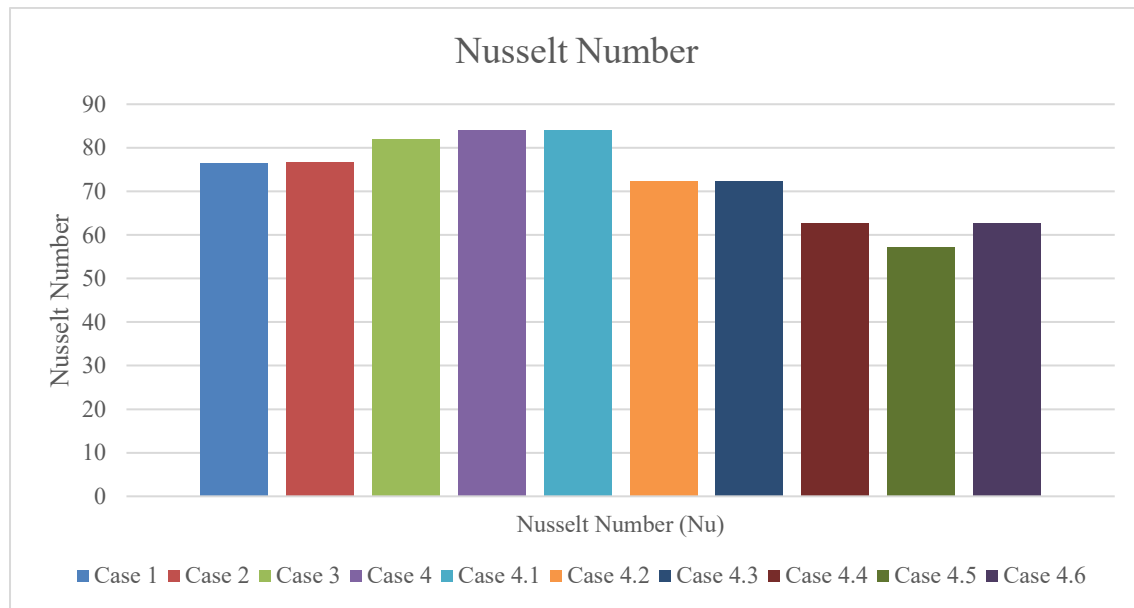


Figure 13: Nusselt number in case 1-4 and case 4.1-4.6

Effect of Fluid Velocity (Both Hot and Cold)

After evaluating the influence of fin geometry, the effect of fluid velocity was investigated using Design 5 (Case 4 configuration) because it provided the highest heat-transfer rate, convective heat-transfer coefficient and Nusselt number among Cases 1–4. Six additional operating conditions, designated Cases 4.1–4.6, were developed by

varying the hot- and cold-fluid mass flow rates. The hot-fluid mass flow rate was varied between 0.13 and 0.20 kg/s, while the cold-fluid mass flow rate was varied between 0.14 and 0.234 kg/s. This analysis was used to determine how changes in flow rate affect the thermal response and hydraulic characteristics of the selected twisted-fin configuration.

Table 5: Outlet temperature of cold and hot fluid in case 4.1-4.6

Parameter	Case 4.1	Case 4.2	Case 4.3	Case 4.4	Case 4.5	Case 4.6
Cold O/L Temp.	300.418	300.79	300.72	301.11	301.179	301.009
Hot O/L Temp.	320.561	320.839	320.421	320.702	321.082	320.147
Pressure Drop Cold	254.308	165.238	165.236	108.971	108.93	109.008
Pressure Drop Hot	11368.5	11368.5	8113.1	8113.1	11368.5	5350.22
Friction factor (F)	0.109	0.12	0.12	0.13	0.13	0.13
Heat Transfer rate (Q) Cold	2219.12	1987.11	1934.11	1732.63	1773.36	1673.87
Heat Transfer rate (Q) Hot	2164.58	1932.84	1882.83	1688.73	1426.43	1632.15
Heat Transfer rate (Q)	2191.85	1959.98	1908.47	1710.68	1599.89	1653.01
Convective Heat transfer	2207.95	1899.34	1900.3	1646.23	1502.67	1647.73
Nusselt Number (Nu)	84.02	72.28	72.318	62.6489	57.1855	62.7

Outlet Temperature

The outlet-temperature results demonstrate that changing the hot- and cold-fluid mass flow rates alters the thermal behaviour of the DPHE. The cold-fluid outlet temperature ranged from 300.418 K in Case 4.1 to 301.179 K in Case 4.5, while the hot-fluid outlet temperature varied from 320.147 K in Case 4.6 to 321.082 K in Case 4.5. Case

4.1 produced a cold-fluid outlet temperature of 300.418 K and a hot-fluid outlet temperature of 320.561 K, whereas Case 4.5 produced the highest outlet temperatures for both streams.

The changes in outlet temperature are governed by the interaction between mass flow rate, fluid residence time and

convective heat transfer. Increasing the flow rate changes the thermal capacity rate of the stream and modifies the time available for heat exchange within the heat exchanger. At higher flow rates, the associated increase in fluid motion can improve convective transport, while the reduced residence

time can limit the temperature change of an individual fluid stream. Consequently, outlet temperature should be interpreted together with the heat-transfer rate rather than as an independent indicator of thermal performance.

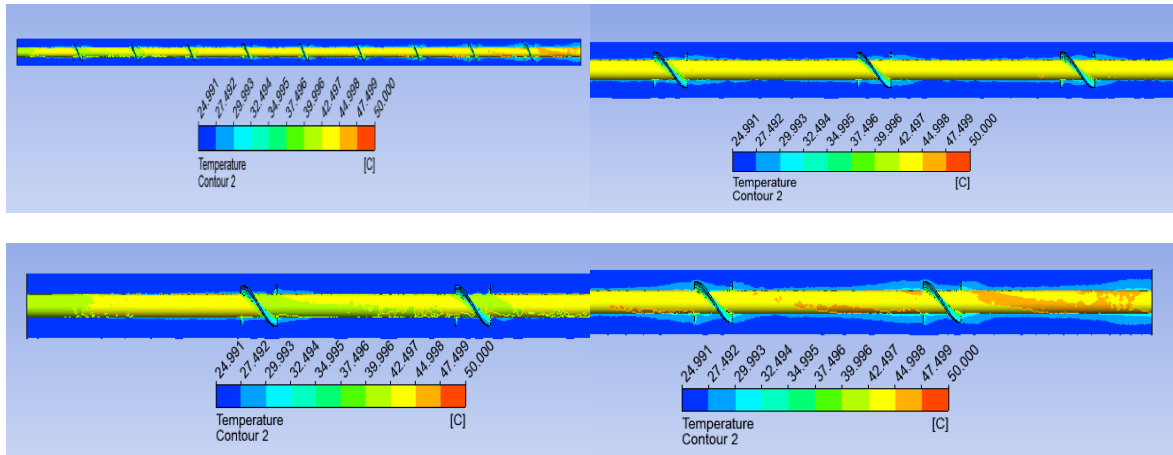


Figure 14: Temperature contour of case 4.1

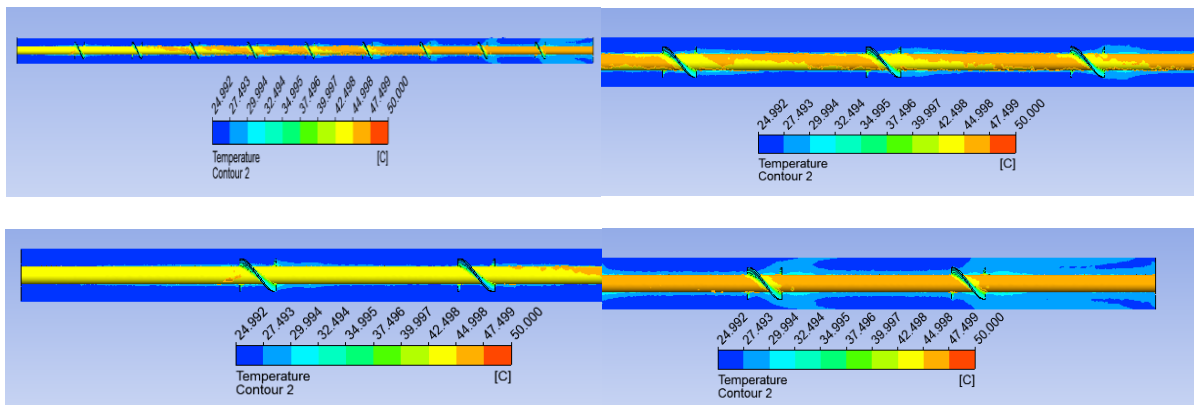


Figure 15: Temperature contour of case 4.5

Pressure Drop and Friction Factor

A substantial variation in pressure drop was observed when the fluid flow rates were changed. The cold-fluid pressure drop decreased from 254.308 Pa in Case 4.1 to approximately 108.93–109.008 Pa in Cases 4.4–4.6. The hot-fluid pressure drop was 11368.5 Pa in Cases 4.1, 4.2 and 4.5, 8113.1 Pa in Cases 4.3 and 4.4, and decreased to 5350.22 Pa in Case 4.6. The reported friction factor increased from 0.109 in Case 4.1 to 0.12 in Cases 4.2 and 4.3, and reached 0.13 in Cases 4.4–4.6.

Pressure drop is directly related to the resistance experienced by the flowing fluid through the finned passage. Changes in mass flow rate modify the velocity field and momentum transport, while the twisted-fin geometry

generates additional disturbance and secondary motion. These effects influence both wall shear stress and local pressure losses. The results indicate that the pressure-drop response is not governed by flow rate alone because the hot- and cold-side mass flow rates were changed simultaneously in different cases. Therefore, the pressure-drop results reflect the combined influence of operating flow conditions and the fixed twisted-fin geometry.

Heat Transfer, Convective Heat Transfer Coefficient and Nusselt Number

The variation in fluid velocity produced a clear change in thermal performance. Case 4.1 recorded the highest average heat-transfer rate of 2191.85 W, followed by 1959.98 W in Case 4.2, 1908.47 W in Case 4.3, 1710.68 W

in Case 4.4, 1599.89 W in Case 4.5, and 1653.01 W in Case 4.6. The convective heat-transfer coefficient followed a similar overall trend, decreasing from 2207.95 W/m²K in Case 4.1 to 1502.67 W/m²K in Case 4.5, before increasing slightly to 1647.73 W/m²K in Case 4.6. The Nusselt number was highest in Case 4.1 at 84.02, followed by 72.28, 72.318, 62.6489, 57.1855 and 62.7 for Cases 4.2–4.6, respectively.

The higher heat-transfer performance obtained in Case 4.1 is associated with its higher cold-fluid mass flow rate and relatively high hot-fluid mass flow rate. Increased fluid movement promotes stronger convection and enhances interaction between the fluid and twisted-fin surfaces. The resulting disturbance of the thermal boundary layer improves convective heat transport, which is reflected in the higher convective heat-transfer coefficient and Nusselt number. As the flow conditions are changed toward the lower-flow-rate cases, the thermal enhancement decreases, resulting in lower values of heat-transfer rate, convective heat-transfer coefficient and Nusselt number. The slight recovery observed in Case 4.6 indicates that the relationship between flow rate and thermal performance is coupled with the simultaneous variation of both hot- and cold-side flow conditions. Overall, the results demonstrate that fluid velocity is a major operating parameter controlling the thermal performance of the twisted-fin DPHE.

CONCLUSION

The present numerical study investigated the thermal and hydraulic performance of a double-pipe heat exchanger equipped with twisted-fin inserts. The study was conducted in two major parts. The first part examined the effect of different twisted-fin geometries on the heat-transfer and flow characteristics of the heat exchanger under the same hot- and cold-fluid flow conditions. The second part investigated the effect of velocity variation of both the hot and cold fluids using the selected fin configuration. The performance of the different cases was evaluated in terms of outlet temperature, pressure drop, friction factor, heat-transfer rate, convective heat-transfer coefficient, and Nusselt number. Based on the comparative results obtained from the numerical analysis, the following conclusions were drawn:

1. Case 4 was identified as the optimized configuration in terms of fin geometry. Among the investigated fin geometries, Case 4 provided the highest overall heat-transfer performance, with an overall heat-transfer rate of 2126.06 W, convective heat-transfer coefficient of 2206.88, and Nusselt

number of 83.985. It also produced the highest cold-fluid outlet temperature of 300.349 K, indicating enhanced thermal interaction between the hot and cold fluids.

2. Case 4.1 provided the highest heat-transfer performance according to heat-transfer rate, convective heat-transfer coefficient, and Nusselt number. The case achieved an overall heat-transfer rate of 2191.85 W, a convective heat-transfer coefficient of 2207.95, and a Nusselt number of 84.02, which were the highest values among the velocity-variation cases.
3. Case 4.5 produced the highest outlet temperatures among the velocity-variation cases. The cold-fluid outlet temperature reached 301.179 K, while the hot-fluid outlet temperature reached 321.082 K. Thus, Case 4.5 showed the highest outlet-temperature values under the investigated combinations of hot- and cold-fluid flow rates.
4. The results also demonstrate that increasing thermal performance is accompanied by changes in hydraulic characteristics. The cases showing enhanced heat transfer generally experienced greater flow resistance, reflected by changes in pressure drop and friction factor. Therefore, the selection of a suitable configuration depends on the desired balance between heat-transfer enhancement and hydraulic performance.
5. Overall, the numerical investigation confirms that twisted-fin geometry and hot- and cold-fluid velocity variation significantly influence the thermal and flow characteristics of the double-pipe heat exchanger. The results provide a basis for selecting the appropriate fin geometry and operating conditions according to the required heat-transfer performance.

FUTURE SCOPE

The present numerical investigation provides a foundation for further enhancement of the thermal and hydraulic performance of double-pipe heat exchangers with twisted-fin inserts. Future studies can investigate the use of nanofluids containing nanoparticles such as Al₂O₃, CuO, TiO₂, SiO₂, or hybrid nanoparticles at different concentrations to evaluate their effects on heat-transfer rate, Nusselt number, and pressure drop. The influence of alternative tube and fin materials with different thermal conductivities can also be examined. Further improvements may be achieved by optimizing fin height, thickness, pitch, twist angle, number of fins, number of turns, spacing, and

arrangement. In addition, other passive heat-transfer enhancement geometries, including wavy, ribbed, serrated, perforated, pin, and corrugated fins, can be investigated and compared. Finally, experimental testing of the selected configuration can be conducted to validate the CFD results and assess its practical thermal-hydraulic performance.

REFERENCES

- Ali, S., Dbouk, T., Wang, G., Wang, D., & Drikakis, D. (2023). Advancing thermal performance through vortex generators morphing. *Scientific Reports*, 13(368), 1–14. <https://doi.org/10.1038/s41598-022-25516-4>
- Aridi, R., Ali, S., Lemenand, T., Faraj, J., & Khaled, M. (2022). CFD analysis on the spatial effect of vortex generators in concentric tube heat exchangers – A comparative study. *International Journal of Thermofluids*, 16, 100247. <https://doi.org/10.1016/j.ijft.2022.100247>
- Arjmandi, H., Amiri, P., & Saffari Pour, M. (2020). Geometric Optimization of a Double Pipe Heat Exchanger with Combined Vortex Generator and Twisted Tape: A CFD and Response Surface Methodology (RSM) Study. *Thermal Science and Engineering Progress*, 18, 100514. <https://doi.org/10.1016/j.tsep.2020.100514>
- Bhattacharjee, D. (2020). CFD Analysis of Double Pipe Counter Flow Heat Exchanger. *International Journal of Engineering Research & Technology*, 9(10).
- Chaudhari, N. R., & Adroja, F. N. (2014). A Review on Design & Analysis of Double Pipe Heat Exchanger. *International Journal of Engineering Research & Technology*, 3(2), 2502–2505.
- Dandoutiya, B. K., & Kumar, A. (2023). Study of thermal performance of double pipe heat exchanger using W-cut twisted tape. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 45(2), 5221–5238. <https://doi.org/10.1080/15567036.2023.2207497>
- El Hassan, M., Al Rajeh, A. Y., Bukharin, N., Alghannam, Y., & Albakawi, J. (2025). Performance Enhancement of Convective Heat Transfer in Double Pipe Heat Exchangers using Different Vortex Generators' Configurations. *E3S Web of Conferences*, 647. <https://doi.org/10.1051/e3sconf/202564702002>
- Gabir, M. M., & Alkhafaji, D. (2021). Comprehensive Review on Double Pipe Heat Exchanger Techniques. *Journal of Physics: Conference Series*, 1973. <https://doi.org/10.1088/1742-6596/1973/1/012013>
- Hameed, V. M., & Awad, S. M. (2021). Effect of fluid velocity on the heat transfer rate of double pipe heat exchanger. *International Journal of Multidisciplinary Sciences and Advanced Technology*, 2(1), 1–9. <http://www.ijmsat.com>
- Haw, W., Oon, C. S., & Lau, E. Von. (2025). Thermal-Hydraulic and Hydrodynamic Analysis of Twisted Fins in a Double-Pipe Heat Exchanger Willy. *Results in Engineering*, 26(March). <https://doi.org/10.1016/j.rineng.2025.104642>
- Kadhim, S. A., Rasheed, R. H., Hammoodi, K. A., Ashour, A. M., Rashid, F. L., Saeed, M. M. A., Bouabidi, A., Shoeibi, S., Aich, W., & Hasan, M. M. (2025). Enhancing the thermal performance of double-pipe heat exchangers using wire coil inserts: A focused review. *Scientific African*, 29, e02896. <https://doi.org/10.1016/j.sciaf.2025.e02896>
- Kotian, S., Methekar, N., Jain, N., & Naik, P. (2020). Heat Transfer and Fluid Flow in a Double Pipe Heat Exchanger, Part I: Experimental Investigation. *Asian Review of Mechanical Engineering*, 9(2).
- Kumbhar, S. S., Patil, P. U. N., Patil, Y. P., Shelake, A. A., Dhere, K. T., & Shinde, P. N. (2023). Cfd Analysis of Double Pipe Heat Exchanger. *International Research Journal of Modernization in Engineering Technology and Science*, 05(06), 123–127. <https://doi.org/10.56726/irjmets41354>
- Lavakumar, M., Reddy, B. V., & Reddy, K. H. (2024). Enhancement of Heat Transfer rate using Axially Interrupted Rectangular Fins in Double Pipe Heat Exchanger. *International Research Journal of Multidisciplinary Technovation*, 6(6).
- Navindgi, M. C. (2018). Heat Transfer Enhancement of Heat Exchanger Using Vortex Generators. 4(7).
- Omidi, M., Farhadi, M., & Jafari, M. (2017). A comprehensive review on double pipe heat exchangers. *Applied Thermal Engineering*, 110, 1075–1090. <https://doi.org/https://doi.org/10.1016/j.applthermaleng.2016.09.027>

- Rai, V. G., & Muchrikar, A. (2021). CFD analysis of double pipe counter flow heat exchanger subjected to varying cold fluid velocity in flat inner pipe. *International Journal of Engineering Development and Research*, 9(4), 37–43.
- Rifqi, A. M., & Al-Khalil, Y. (2025). Thermal Performance Analysis of Heat Exchanger Using Computational Fluid Dynamics. *RESWARA: Jurnal Riset Ilmu Teknik*, 3(2), 60–68. <https://doi.org/10.70716/reswara.v3i2.416>
- Rostami, S., & Ahmadi, N. (2025). In the Study of the Effects of the Pipe Design of a Heat Exchanger on the Thermo-Fluid Characteristics and Exergy Destruction. *Processes*, 13(835).
- Savas, Evran, & Kurt, M. (2023). CFD and statistical analysis of flow mass ratios on temperature and pressure drops of double pipe heat exchanger. *International Journal of Low-Carbon Technologies*, 18.
- Singh, S. K., Chauhan, M. K., & Shukla, A. K. (2022). Heat transfer enhancement in double-pipe heat exchanger: A review. *Journal of Physics: Conference Series*, 2178. <https://doi.org/10.1088/1742-6596/2178/1/012007>
- Talakonda, Y., Jayachandraiah, B., & Reddy, B. (2019). Performance Enhancement of Double Pipe Heat Exchanger with Helical Fin and Vortex Generator using CFD. *International Journal of Engineering and Advanced Technology*, 9, 2677–2681. <https://doi.org/10.35940/ijeat.B4058.129219>
- Usmani, A. A., Shrivastava, P., & Singh, S. (2025). Impacts of Fins on The Heat Transfer Characteristics in A Shell and Cone-Shaped Coil Heat Exchanger. *International Journal of Innovations In Science Engineering And Management*, 4(1). <https://doi.org/10.69968/ijisem.2025v4i1106-117>