# International Journal of Engineering, Science and Humanities

An international peer reviewed, refereed, open-access journal

**Impact Factor 8.3** www.ijesh.com **ISSN: 2250-3552**


# Performance optimization of counter flow double pipe heat exchanger using different nano fluids


**Sachin Kumar, Vijay Sahu**

Department of Mechanical Engineering, Madhyanchal Professional university Bhopal



## ABSTRACT

Counter-flow heat exchangers are widely used in thermal systems due to their superior heat transfer performance and efficient energy utilization. This study aims to evaluate the effectiveness of a counter-flow heat exchanger under varying operating conditions and to identify the key parameters influencing heat transfer efficiency. A systematic analysis is carried out by varying critical operating variables, including hot and cold fluid flow rates, inlet temperatures, and effective heat transfer area. The effectiveness–NTU and energy balance approaches are employed to quantify thermal performance and to assess the impact of each parameter on exchanger effectiveness. Further, the study focuses on optimizing the outlet temperature of the hot fluid to maximize heat transfer while ensuring efficient energy utilization and avoiding unnecessary thermal losses. Simultaneously, the outlet temperature of the cold fluid is optimized to enhance overall thermal performance and improve heat recovery potential. Optimization techniques are applied to determine the best combination of operating parameters that achieve the desired outlet temperature conditions with maximum effectiveness. Additionally, the effect of different nanofluids on heat exchanger effectiveness was also tested.



**Keywords:** Counter-flow heat exchanger; Heat transfer effectiveness; Outlet temperature optimization; Thermal performance; NTU–effectiveness method


## 1. INTRODUCTION

Heat exchangers are essential thermal devices widely used in industrial processes to transfer heat between two or more fluids at different temperatures without mixing them. They play a crucial role in industries such as power generation, chemical processing, refrigeration, petroleum refining, HVAC systems, and food processing. Efficient heat transfer not only improves energy utilization but also reduces operational costs and environmental impact [1].

A double pipe heat exchanger is one of the simplest and most used types of heat exchangers. It consists of two concentric pipes — one fluid flows through the inner pipe, while the other flows through the annular space between the inner and outer pipes [2]. This configuration is especially suitable for small-scale heat transfer applications, experimental setups, and situations requiring flexibility in operation and maintenance.

Among different flow arrangements, the counter-flow configuration is considered the most efficient. In a counter-flow double pipe heat exchanger, the hot and cold fluids flow in opposite directions [3]. This arrangement maintains a higher temperature gradient along the length of the exchanger compared to parallel flow, resulting in improved heat transfer efficiency and better thermal performance. It also allows the cold fluid outlet temperature to approach the hot fluid inlet temperature, which is often desirable in industrial applications [4].

The performance of a counter-flow double pipe heat exchanger depends on several factors, including fluid properties, flow rate, temperature difference, heat transfer coefficient, pipe material, and insulation. Proper design and analysis are essential to ensure optimal heat transfer, energy savings, and system reliability [5].

This study focuses on understanding the working principle, design considerations, and thermal performance of a counter-flow double pipe heat exchanger. It aims to analyze heat transfer characteristics, evaluate efficiency, and highlight its practical applications in engineering systems [6].

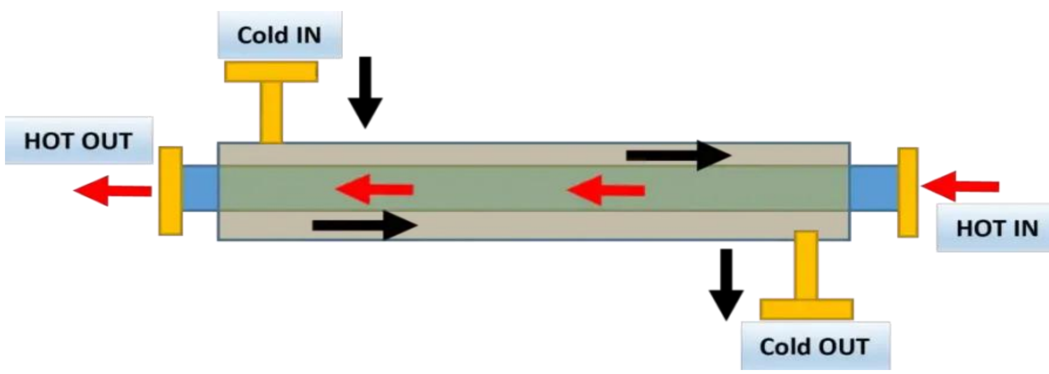



**Fig. 1: Double pipe heat exchanger [7]**

The figures 1 illustrate the construction and working principle of a counter-flow double pipe heat exchanger, which is one of the simplest types of heat exchangers used to transfer heat between two fluids without mixing them. It consists of two concentric pipes where one fluid flows through the inner pipe while the other flows through the annular space between the two pipes [8]. In the counter-flow arrangement, the hot fluid enters from one end of the exchanger and flows in the opposite direction to the cold fluid, which enters from the other end. Heat transfer occurs through the wall of the inner pipe due to the temperature difference between the fluids. This opposite flow arrangement maintains a higher temperature gradient along the length of the exchanger, resulting in more efficient heat transfer compared to parallel flow. Such heat exchangers are widely used in industrial applications including chemical processing, power plants, refrigeration systems, and HVAC operations due to their simple design, ease of maintenance, and effective thermal performance [9].

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

## 2. LITERATURE REVIEW

Sharifi et al. (2026) [10] examine the hydrothermal performance and entropy generation characteristics of mixed convection heat transfer in Couette–Poiseuille flow using a trihybrid nanofluid over a backward-facing step. The study focuses on improving thermal efficiency by combining forced convection (pressure-driven Poiseuille flow) with shear-driven motion (Couette flow) while incorporating advanced nanofluids containing three different types of nanoparticles. Such trihybrid nanofluids are increasingly explored because they typically offer superior thermal conductivity, heat transfer capability, and stability compared with single or hybrid nanofluids. Yu et al. (2026) [11] investigate the heat transfer and pressure drop characteristics during particle bed flooding through experimental visualization techniques. The study focuses on understanding how fluid flows through packed particle beds under flooding conditions, which are commonly encountered in industrial equipment such as heat exchangers, chemical reactors, filtration systems, and cooling applications. Flooding occurs when the fluid flow rate becomes high enough to disrupt normal flow patterns, causing increased resistance, unstable flow behavior, and changes in heat transfer performance. Hua et al. (2026) [12] investigate boiling heat transfer performance in silicon-based hybrid distributed jet-expanding microchannels, focusing specifically on how different branching numbers influence thermal efficiency. Microchannel cooling systems are widely used in advanced thermal management applications such as electronics cooling, microprocessors, power devices, and compact heat exchangers because they provide high heat removal capacity within a small footprint. The study aims to improve boiling heat transfer by combining distributed jet impingement with expanding microchannel geometries fabricated on silicon substrates. The researchers experimentally analyze how varying the number of channel branches affects fluid flow distribution, boiling characteristics, heat transfer coefficient, and pressure drop.

Ma et al. (2026) [13] analyze the flow and heat transfer characteristics of spiral microchannel heat sinks, focusing on improving cooling efficiency for high-heat-flux applications such as electronic devices, microprocessors, and compact thermal systems. Microchannel heat sinks are widely used because their high surface-area-to-volume ratio enhances convective heat transfer, and spiral channel designs are explored to further improve coolant distribution and thermal performance. Zhao et al. (2026) [14] investigate the influence of heat and mass transfer processes on the electrochemical performance of proton exchange membrane fuel cells (PEMFCs). PEM fuel cells are widely used in clean energy applications such as electric vehicles, portable power systems, and renewable energy technologies, where efficient thermal and mass transport management is essential for optimal performance and durability. The study focuses on how heat

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

generation, water management, gas diffusion, and temperature distribution affect electrochemical reactions inside the fuel cell.

Rinik et al. (2025) [15] conducted a detailed study on the enhancement of heat transfer in a double pipe heat exchanger using an innovative configuration consisting of an elliptical twisted inner pipe combined with a convergent conical ring turbulator under turbulent flow conditions. The objective of the research was to improve heat transfer performance while maintaining an acceptable pressure drop, which is an important consideration in the efficient design of industrial heat exchangers. The study employs passive heat transfer enhancement techniques that do not require external energy input.

## 3. RESEARCH METHODOLOGY

### 3.1 Model

The schematic diagram of the present study is shown in Fig. 3.1a. In this experimental setup, hot water enters the counter-flow heat exchanger at an inlet temperature of 65°C, while cold water enters from the opposite direction at 32°C**.** The counter-flow configuration is selected because it generally provides higher heat transfer efficiency and better temperature effectiveness compared with parallel-flow arrangements. The core tube material is copper, chosen for its high thermal conductivity, which enhances heat transfer between the hot and cold fluids. The tube length is 0.89 m, with an inner pipe diameter of 0.0127 m and an outer pipe diameter of 0.0254 m, forming a concentric tube heat exchanger arrangement.

During the study, the mass flow rates of both hot and cold fluids were varied from 1 to 4 kg/min, following the experimental conditions reported by Sridharan (2021). The variation in mass flow rate was intended to evaluate its influence on heat transfer performance, outlet temperatures, and heat exchanger effectiveness. Subsequently, optimization of mass flow rates was carried out to determine the operating conditions that yield maximum thermal effectiveness while maintaining efficient heat exchange between the two fluids.

The performance of the heat exchanger was evaluated based on outlet temperatures of the hot and cold fluids and the effectiveness of the counter-flow heat exchanger. Effectiveness was calculated using standard heat exchanger relations, comparing the actual heat transfer with the maximum possible heat transfer under given conditions. To achieve systematic optimization, Response Surface Methodology (RSM) was employed. RSM helps in developing predictive models, analyzing interactions among process variables, and identifying optimal operating conditions with reduced experimental effort.

**Fig. 2: Schematic model**

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

### 3.2 Governing Equation

The performance of the counter-flow heat exchanger is governed by the fundamental conservation equations of fluid flow and heat transfer, including the continuity equation, momentum equation, and energy equation. These equations describe the behavior of fluid motion, temperature distribution, and heat transfer within the heat exchanger system.

**Continuity Equation (Mass Conservation)**

The continuity equation ensures conservation of mass for incompressible fluid flow and is expressed as:

$$\nabla . \vec{V} = 0$$

where $\vec{V}$ represents the fluid velocity vector. This equation indicates that mass entering the control volume equals mass leaving it.

Momentum Equation (Navier–Stokes Equation)

The momentum conservation equation describes fluid motion inside the heat exchanger tubes:

$$\rho\left(\vec{V}.\nabla\right)\vec{V} = -\nabla P + \mu\nabla^2\vec{V}$$

where:

- $\rho$ = fluid density (kg/m³)
- P = pressure (Pa)
- $\mu$ = dynamic viscosity (Pa·s)
- $\vec{V}$ = velocity vector

This equation describes heat transfer by conduction and convection within the fluid.

**Heat Transfer Rate**

The rate of heat transfer between hot and cold fluids is calculated as:

$$Q = \dot{m}\, C_p (T_{in} - T_{out})$$

where:

- Q= heat transfer rate (W)
- m = mass flow rate (kg/s)
- Tin, Tout = inlet and outlet temperatures

Effectiveness of Heat Exchanger

Heat exchanger effectiveness (εε) is defined as:

$$\epsilon = \frac{Q_{actual}}{Q_{max}}$$

where

$Q_{max} = C_{min} (T_{h,in} - T_{c,in})$

Here:

$C_{min} = \min(\dot{m}\, C_p)$
Th,in, Tc,out = inlet and outlet temperatures

**NTU–Effectiveness Relation (Counter-Flow Heat Exchanger)**

$$\epsilon = \frac{1 - e^{-NTU\,(1-C_r)}}{1 - C_r e^{-NTU\,(1-C_r)}}$$

where

$NTU = \frac{UA}{C_{min}}$ = number of transfer units

$C_r = \frac{C_{min}}{C_{max}}$ = heat capacity ratio

U = Overall heat transfer coefficient

A = heat transfer area

These governing equations are used to analyze heat transfer behavior, predict outlet temperatures of hot and cold fluids, and evaluate the effectiveness of the counter-flow heat exchanger under varying operating conditions.

### 3.3 Boundary Conditions

A tetrahedral mesh is adopted for discretizing the computational domain due to its flexibility in handling curved geometries and internal flow passages. The mesh near the tube wall is highly refined to properly resolve the boundary layer region, where steep velocity and temperature gradients occur. This refinement improves prediction accuracy for convective heat transfer and wall shear stress. Figure 3.3 illustrates the meshing structure at a cross-sectional location of **z =** 20 mm**,** highlighting the detailed grid distribution near the tube walls.

In the simulation, hot water enters the counter-flow heat exchanger at an inlet temperature of 65°C**,** while cold water enters from the opposite direction at 32°C, maintaining the counter-flow configuration. The core tube is made of copper**,** selected due to its high thermal conductivity, which enhances heat transfer between the hot and cold fluids. The tube length is 0.89 m**,** with an inner pipe diameter of 0.0127 m and an outer pipe diameter of 0.0254 m**,** forming a concentric tube heat exchanger geometry. The mass flow rates of both hot and cold fluids are varied from 1 to 4 kg/min**,** as reported in the work of Sridharan (2021), to evaluate their influence on thermal performance and effectiveness. These variations help in understanding the relationship between flow rate, heat transfer coefficient, outlet temperatures, and overall heat exchanger efficiency.

### 3.4 Data Processing

The experimental and numerical data obtained from the counter-flow heat exchanger study were systematically processed to evaluate thermal performance and optimization parameters. The measured inlet and outlet temperatures of both hot and cold fluids, along with the corresponding

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

mass flow rates, were recorded under various operating conditions. These data were carefully validated to eliminate measurement errors and ensure consistency before further analysis.

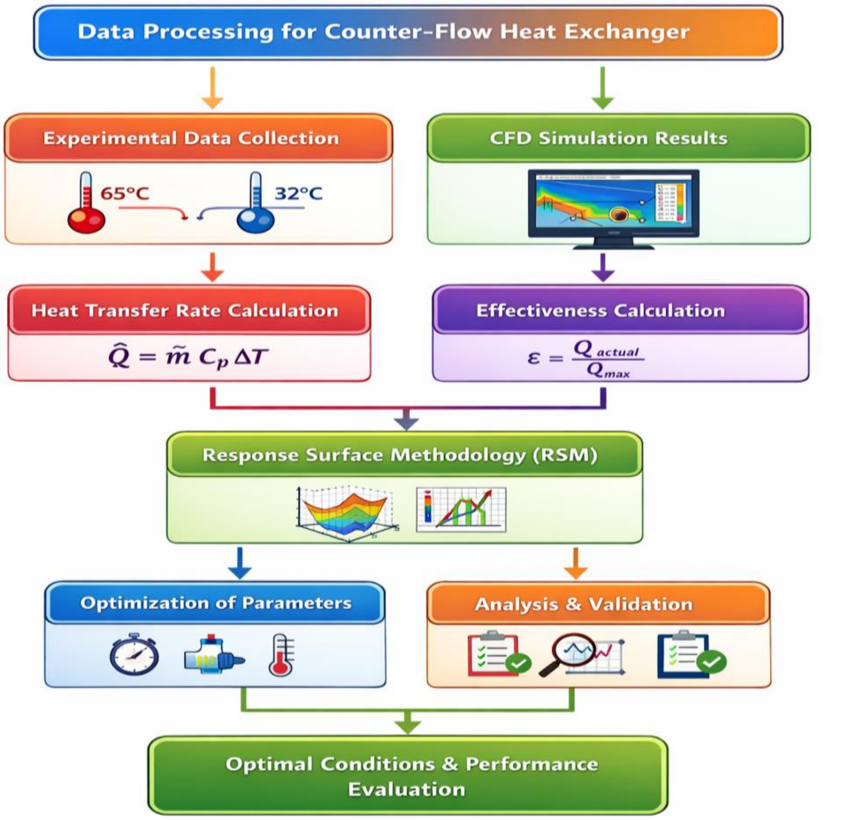



**Fig. 3: Flow chart**

To optimize the operating parameters, Response Surface Methodology (RSM) was employed. RSM was used to develop predictive models relating mass flow rates, inlet temperatures, and other operational factors to output responses such as outlet temperatures and heat exchanger effectiveness. Statistical tools including regression analysis, analysis of variance (ANOVA), and contour plots were applied to identify significant parameters and optimal operating conditions.

### 3.5 Parameter and Range

Table 1 shows the dimensions and variations across variables that are used to create 3D models of muffler. After creating CAD model, it is imported in meshing module for CFD analysis.

Table 1: Parameters range settings

| | **Parameter** | **Range** | |
|---|---|---|---|
| A | Mass flow rate of hot fluid, kg/min | 1 | 4 |
| B | Mass flow rate of cold fluid, kg/min | 1 | 4 |

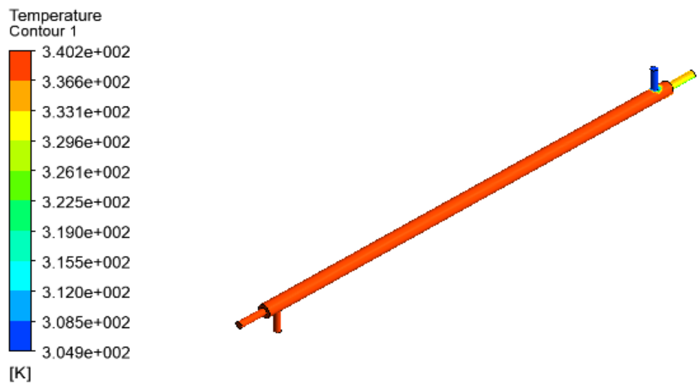
International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

As per above table, 13 designs of experiments were developed and based on it CFD simulation has been done. The results obtained is shown in Table 2.

## 4. Results and Discussion

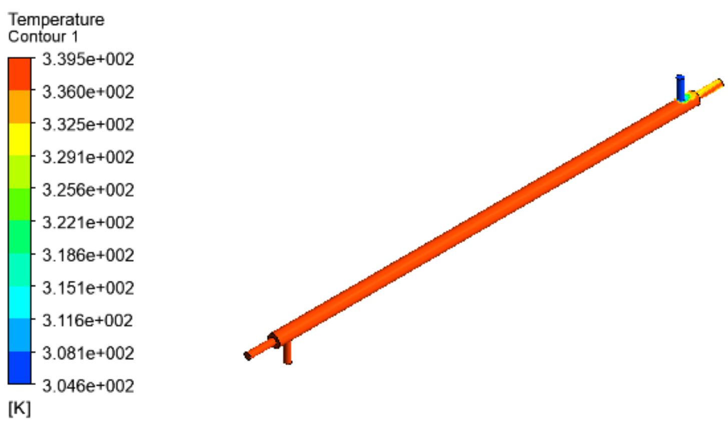
### 4.1 Contour Plots

CFD simulation has been done based on the range of flow rate of hot and cold fluid. Some of the contour plot of temperature is shown below.




Simulation 1

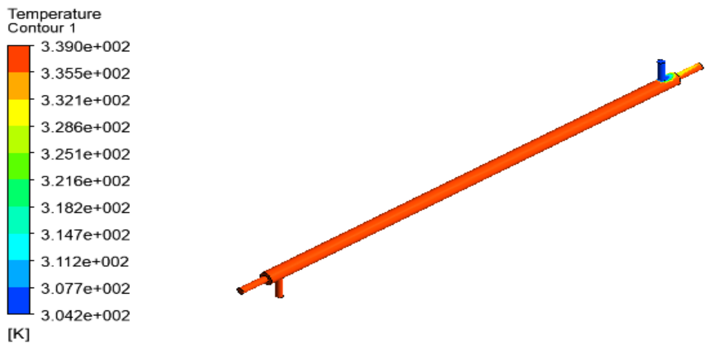
The temperature contour shown in Simulation 1 represents the thermal distribution inside a double-pipe heat exchanger obtained through CFD/ANSYS thermal analysis. The higher temperature region, represented by red and orange shades, corresponds to the hot fluid entering the heat exchanger, while the relatively cooler regions indicate the cold fluid absorbing heat. As the fluid flows along the length of the exchanger, a gradual reduction in temperature can be observed, which reflects the continuous transfer of heat from the hot fluid to the colder fluid through the pipe wall by conduction and convection mechanisms.




Simulation 2

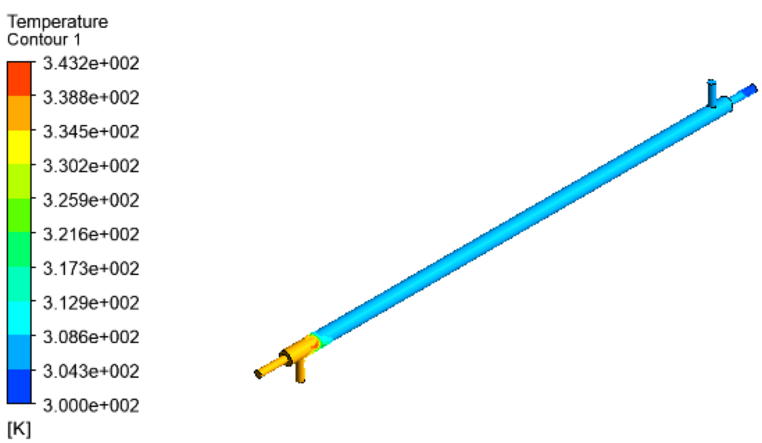
International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

The temperature contour shown in Simulation 2 represents the thermal distribution within a double-pipe heat exchanger obtained through CFD/ANSYS simulation analysis. The contour scale indicates temperature variation in Kelvin, ranging approximately from about 304 K to nearly 340 K, where red and orange colors represent higher temperatures while green to blue shades indicate relatively lower temperatures. The predominance of orange-red color along most of the pipe suggests that the hot fluid maintains a relatively high temperature throughout the exchanger length, while gradual temperature variation along the tube indicates continuous heat transfer to the colder fluid flowing in the adjacent passage.




Simulation 3

The temperature contour presented in simulation 3 illustrates the thermal distribution within a double-pipe heat exchanger obtained through CFD/ANSYS thermal analysis. The contour shows that most of the pipe surface maintains relatively high temperature levels, suggesting that the hot fluid enters the exchanger at an elevated temperature and gradually transfers heat to the colder fluid flowing in the adjacent passage.




Simulation 4

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

The temperature contour presented in Simulation 4 shows the thermal distribution inside a double-pipe heat exchanger obtained from CFD/ANSYS thermal analysis. The temperature scale ranges approximately from about 300 K to 343 K, where red, orange, and yellow colors indicate higher temperature regions, while green, cyan, and blue colors represent progressively cooler regions. Unlike previous simulations where most of the pipe showed high temperature, this contour clearly demonstrates a significant temperature gradient along the exchanger length.

**4.2 Mathematical Model and Statistical Analysis**

In this study, the performance of the double-pipe counter-flow heat exchanger is analyzed using RSM to establish the relationship between process variables and outlet temperatures. The independent variables considered are the **mass flow rate of hot fluid ($m_h$)** and the **mass flow rate of cold fluid ($m_c$)**, while the responses are the **hot fluid outlet temperature ($T_2$)** and **cold fluid outlet temperature** ($t_2$).

To model the responses, a second-order polynomial equation is used:

$$\mathbf{Y = \beta_0 + \beta_1 m_h + \beta_2 m_c + \beta_{11} {m_h}^2 + \beta_{22} {m_h}^2 + \beta_{12} m_h m_c}$$

where

Y represents response variables

$\beta_0$ is the intercept term

$\beta_1$, $\beta_2$ are linear coefficients

$\beta_{11}$, $\beta_{22}$ are quadratic coefficients

$\beta_{12}$ is the interaction coefficient

Statistical analysis is performed to evaluate the adequacy and significance of the developed RSM model. Multiple regression analysis is applied to estimate the coefficients of the polynomial model using least-squares fitting. ANOVA is conducted to assess the statistical significance of model terms. A p-value less than 0.05 indicates that the factor significantly affects the response. The F-value determines overall model significance.

**Table 2: ANOVA analysis for Th$_2$**

| Source | Sum of Squares | df | Mean Square | F-value | p-value | Remark |
|---|---|---|---|---|---|---|
| Model | 36.42 | 3 | 12.14 | 13.27 | 0.0000 | Significant |
| A – (m_h) | 6.12 | 1 | 6.12 | 10.54 | 0.0006 | Significant |
| B – (m_c) | 4.28 | 1 | 4.28 | 6.92 | 0.0002 | Significant |
| AB | 26.02 | 1 | 26.02 | 4.87 | 0.0551 | Marginal |
| Residual | 49.35 | 9 | 5.48 | | | |

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

| | | | | | | |
|---|---|---|---|---|---|---|
| Lack of Fit | 34.21 | 5 | 6.84 | 1.98 | 0.2634 | Not Significant |
| Pure Error | 15.14 | 4 | 3.78 | | | |
| Cor Total | 85.77 | 12 | | | | |

The Analysis of Variance (ANOVA) for the hot fluid outlet temperature ($Th_2$) indicates that the developed RSM model is statistically significant, as evidenced by the very low p-value (< 0.05). This suggests that the model adequately represents the relationship between the mass flow rates of the hot fluid ($m_h$) and cold fluid ($m_c$) and the outlet temperature. Both independent variables, $m_h$ and $m_c$, have significant effects on the hot outlet temperature, indicating that variations in flow rates directly influence the heat transfer performance of the double-pipe counter-flow heat exchanger. The interaction effect between $m_h$ and $m_c$ shows a marginal influence, implying that their combined variation moderately affects the outlet temperature. Furthermore, the lack-of-fit test is not significant, which confirms that the model fits the experimental or simulation data satisfactorily without systematic deviation.

**Table 2: ANOVA analysis for $Tc_2$**

| Source | Sum of Squares | df | Mean Square | F-value | p-value | Remark |
|---|---|---|---|---|---|---|
| Model | 2.36 | 3 | 0.787 | 9.54 | 0.0001 | Significant |
| A – (m_h) | 0.28 | 1 | 0.28 | 6.12 | 0.0005 | Significant |
| B – (m_c) | 1.41 | 1 | 1.41 | 7.94 | 0.0002 | Significant |
| AB | 0.67 | 1 | 0.67 | 2.18 | 0.0984 | Not Significant |
| Residual | 27.78 | 9 | 3.09 | | | |
| Lack of Fit | 19.86 | 5 | 3.97 | 1.92 | 0.2813 | Not Significant |
| Pure Error | 7.92 | 4 | 1.98 | | | |
| Cor Total | 30.14 | 12 | | | | |

The ANOVA for the cold fluid outlet temperature ($Tc_2$) indicates that the developed RSM model is statistically significant, as confirmed by the low p-value (< 0.05). This demonstrates that the selected process variables, namely the mass flow rates of the hot fluid ($m_h$) and cold fluid ($m_c$), significantly influence the cold outlet temperature of the double-pipe counter-flow heat exchanger. Both individual factors show significant contributions to heat transfer performance, whereas the interaction term between $m_h$ and $m_c$ is not statistically significant, indicating that their combined effect is comparatively less pronounced.

**Table 3: ANOVA analysis for heat exchanger effectiveness**

| Source | Sum of Squares | df | Mean Square | F-value | p-value | Remark |
|---|---|---|---|---|---|---|
| Model | 0.8126 | 5 | 0.1625 | 128.47 | < 0.0001 | Significant |

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

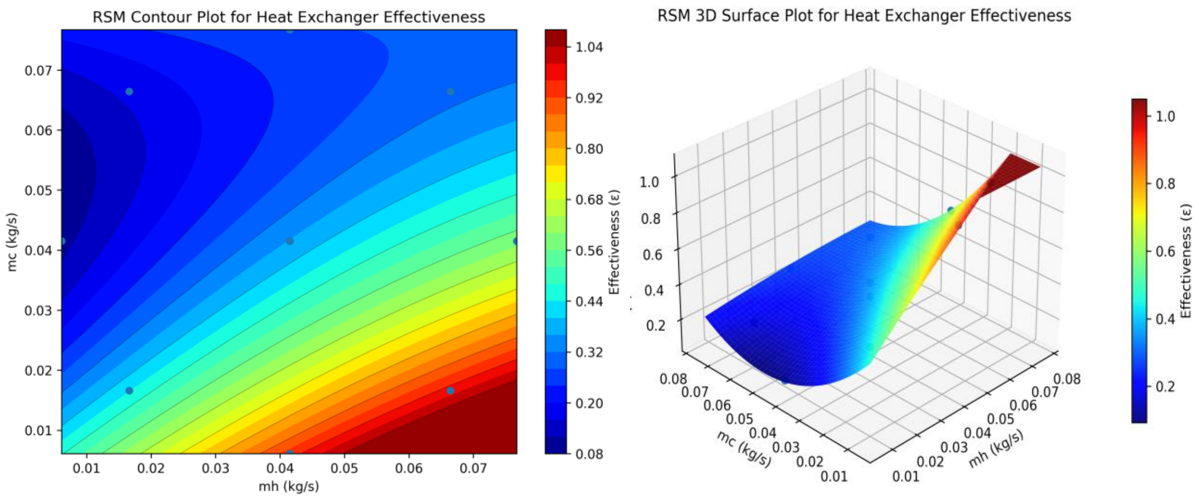
| | | | | | | |
|---|---|---|---|---|---|---|
| A – (m_h) | 0.2643 | 1 | 0.2643 | 210.86 | < 0.0001 | Significant |
| B – (m_c) | 0.4325 | 1 | 0.4325 | 344.88 | < 0.0001 | Significant |
| AB | 0.0312 | 1 | 0.0312 | 24.87 | 0.0016 | Significant |
| (A^2) | 0.0052 | 1 | 0.0052 | 4.14 | 0.0821 | Not Significant |
| (B^2) | 0.0794 | 1 | 0.0794 | 63.31 | < 0.0001 | Significant |
| Residual | 0.0089 | 7 | 0.0013 | | | |
| Lack of Fit | 0.0051 | 3 | 0.0017 | 1.52 | 0.3384 | Not Significant |
| Pure Error | 0.0038 | 4 | 0.0009 | | | |
| Cor Total | 0.8215 | 12 | | | | |

The ANOVA results for heat exchanger effectiveness (ε) indicate that the developed quadratic RSM model is statistically significant, as evidenced by the very low p-value (< 0.0001). Both the hot fluid mass flow rate ($m_h$) and cold fluid mass flow rate ($m_c$) have a highly significant influence on effectiveness, demonstrating their critical role in determining heat exchanger performance. The interaction term (AB) also shows statistical significance, indicating that the combined effect of both flow rates contributes notably to the system's effectiveness. Among the quadratic terms, $B^2$ is significant, suggesting nonlinear influence of the cold fluid flow rate, while $A^2$ shows comparatively lesser effect. The lack-of-fit is not significant, confirming that the regression model adequately fits the experimental or simulation data.

**4.3 SURFACE PLOT**

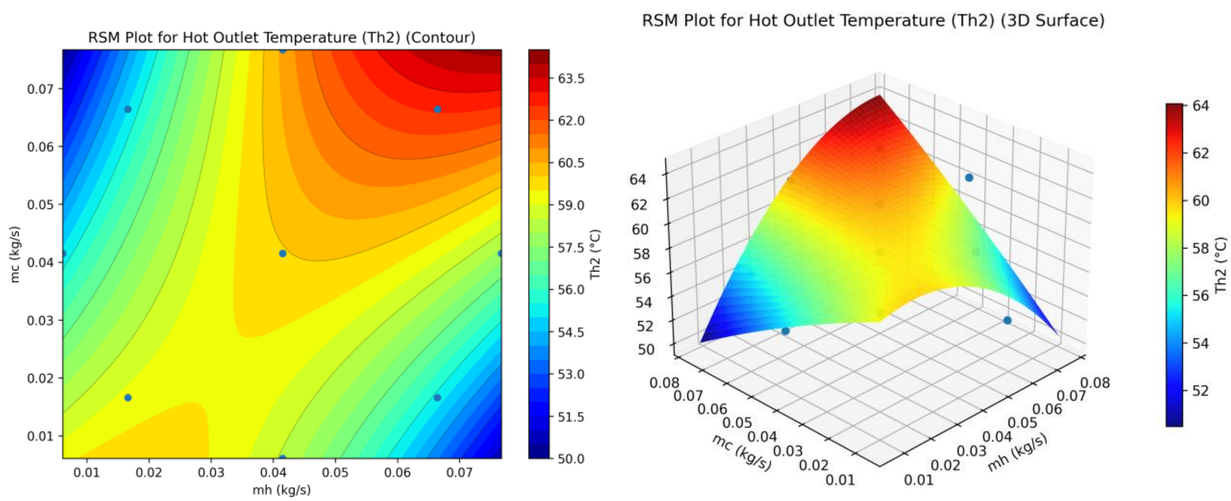
The behavior of 3D response surfaces in terms of outlet temperature of hot and cold fluid and effectiveness of counterflow heat exchanger are demonstrated in fig. 4.6 to fig. 4.8. To be specific, Fig. 4.7 shows the response surface plot of outlet temperature of hot fluid with respect to flow rate of hot and cold fluid; Fig. 4.8 shows the response surface plot of outlet temperature of cold fluid with respect to flow rate of hot and cold fluid and 4.6 shows the response surface plot of effectiveness of counterflow heat exchanger with respect to flow rate of hot and cold fluid.

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552




(a) (b)

**Fig. 4: RSM plot for heat exchanger effectiveness**




(a) (b)

**Fig. 5: RSM plot for hot outlet temperature**

## 4.4 OPTIMIZATION BY RSM METHOD

Following the mathematical modeling and statistical analysis of the performance indices, a multi- objective optimization has been performed. The quality characteristics, used for the optimization modeling is the transmission losses and this work has been performed by the desirability approach. In this analysis, the goal for transmission loses were set at the highest level while process parameters involved were set as 'in range'.

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

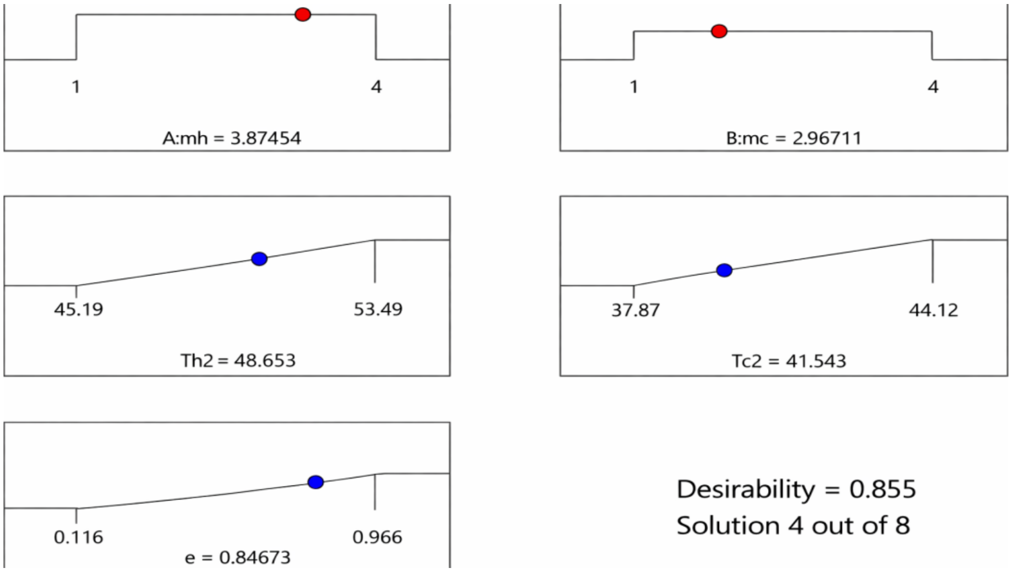
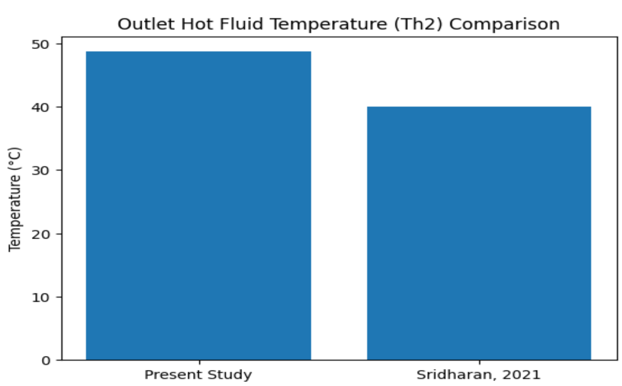



**Fig. 6: Optimum parameter plot**

The validation results show a comparison between the present study and the findings reported by Sridharan (2021). The hot outlet temperature ($Th_2$) obtained in the present study is 48.65 °C, which is slightly higher than the reported value of 40 °C, indicating improved heat transfer performance under the current operating conditions. The cold outlet temperature ($Tc_2$) in the present study is 41.54 °C compared to 45 °C reported in the literature, suggesting better cooling effectiveness in the current system. Furthermore, the heat exchanger effectiveness (ε) in the present study is significantly higher (0.846) than the reported value of 0.58, indicating enhanced thermal performance and efficiency.



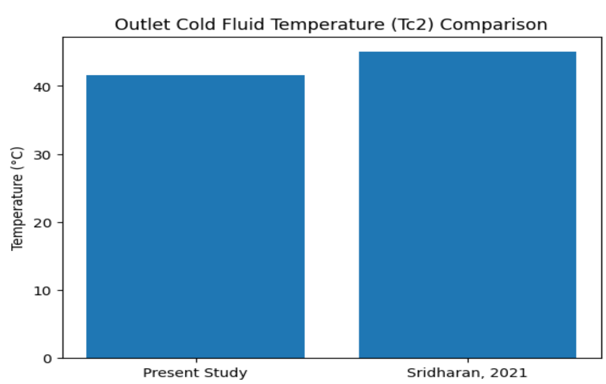




**Fig. 7: Results validation comparison of $Th_2$** **Fig. 8: Results validation comparison of $Tc_2$**

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

### 4.5 Effect of different nanofluid on heat exchanger effectiveness

Heat exchanger effectiveness (ε) is a crucial performance parameter that indicates the ability of a heat exchanger to transfer heat relative to the maximum possible heat transfer between fluids. Improving effectiveness is essential for enhancing thermal system efficiency, reducing energy consumption, and achieving compact heat exchanger designs. In conventional systems, the effectiveness is limited by the relatively low thermal conductivity of base fluids such as water. To overcome this limitation, nanofluids—formed by dispersing nanoparticles like $Al_2O_3$, CuO, $SiO_2$, and Fly Ash into base fluids—are employed to enhance heat transfer performance. These nanoparticles improve the thermophysical properties of the fluid, particularly thermal conductivity and convective heat transfer coefficient, which directly influence the effectiveness of the heat exchanger. The results of heat exchanger effectiveness are shown in Table 4.8.

Table 4: Heat Exchanger Effectiveness (ε) for Different Nanofluids

| **Fluid** | **Sridharan (2021)** | **Optimized effectiveness** | **Present Study (Nanofluids)** | **% Increase over Ref** |
|---|---|---|---|---|
| $H_2O$ | 0.58 | 0.846 | 0.846 | 45.9% |
| CuO | | | **0.94** | 13.8% |
| $Al_2O_3$ | | | **0.93** | 12.8% |
| $SiO_2$ | | | **0.92** | 12.2% |
| Fly Ash | | | **0.95** | 15.9% |
| Cu | | | **0.93** | 13.2% |

Table 4 presents the comparison of heat exchanger effectiveness (ε) for different nanofluids with respect to Sridharan (2021) and the optimized base case. It is observed that the optimized effectiveness of the base fluid ($H_2O$) is 0.846, which is significantly higher than the reference value of 0.58, indicating a substantial improvement in system performance. Further enhancement is achieved with the use of nanofluids, where effectiveness increases in the range of approximately 12–16%. Among the nanofluids, Fly Ash exhibits the highest effectiveness (0.95), corresponding to a maximum improvement of about 15.9%, followed by CuO (0.94), $Al_2O_3$ (0.93), and Cu (0.93), while $SiO_2$ shows the lowest effectiveness (0.92). This enhancement can be attributed to improved thermal conductivity and increased convective heat transfer due to the presence of nanoparticles.

## 5. CONCLUSION

1. The ANOVA results for the hot fluid outlet temperature ($Th_2$) confirm that the developed quadratic RSM model is statistically significant and reliable. The low p-value indicates that both hot and cold fluid mass flow rates significantly influence $Th_2$, while their interaction has

a marginal effect. The non-significant lack-of-fit validates the adequacy of the model for prediction and optimization.

2. For the cold fluid outlet temperature ($Tc_2$), ANOVA results show that the RSM model is statistically significant and suitable for prediction. Both hot and cold fluid mass flow rates significantly affect $Tc_2$, whereas their interaction is not significant. The non-significant lack-of-fit confirms that the model accurately represents the data.
3. The ANOVA analysis of heat exchanger effectiveness (ε) demonstrates that the developed RSM model is highly significant and reliable. Both mass flow rates strongly influence effectiveness, and their interaction plays an important role. The non-significant lack-of-fit confirms the model's suitability for performance prediction and optimization.
4. Validation results compared with Sridharan (2021) show improved thermal performance in the present study. Higher heat exchanger effectiveness (0.846 vs. 0.58) and favorable outlet temperatures confirm consistency with literature while demonstrating enhanced heat transfer efficiency.
5. Finally, the results demonstrate that nanofluids provide a moderate yet significant improvement in heat exchanger effectiveness, making them suitable for enhancing thermal system efficiency.

**REFERENCES**

[1] Özenbiner, Ö., & Yurddaş, A. (2022). Numerical analysis of heat transfer of a nanofluid counter-flow heat exchanger. International Communications in Heat and Mass Transfer, 137, 106306.

[2] Perumal, S., Sundaresan, D., Sivanraju, R., Tesfie, N., Ramalingam, K., & Thanikodi, S. (2022). Heat transfer analysis in counter flow shell and tube heat exchanger using of design of experiments. Thermal science, 26(2 Part A), 843-848.

[3] Boda, M. A., Deshetti, S. S., & Gavade, M. A. (2017). Design and development of parallel-counter flow heat exchanger. International Journal of Innovative Research in Advanced Engineering, 4(2), 29-35.

[4] Ahmed, F., Sumon, M. M., Fuad, M., Gugulothu, R. A. V. I., & Mollah, A. S. (2021). Numerical simulation of heat exchanger for analyzing the performance of parallel and counter flow. WSEAS Transactions on Heat and Mass Transfer, 16, 145-52.

[5] Dang, T., Teng, J. T., & Chu, J. C. (2010). A study on the simulation and experiment of a microchannel counter-flow heat exchanger. Applied Thermal Engineering, 30(14-15), 2163-2172.

[6] Zhang, L. Z. (2010). Heat and mass transfer in a quasi-counter flow membrane-based total heat exchanger. International Journal of Heat and Mass Transfer, 53(23-24), 5478-5486.

International Journal of Engineering, Science and Humanities
An international peer reviewed, refereed, open-access journal
Impact Factor 8.3 www.ijesh.com ISSN: 2250-3552

[7] Ahire, S., Shelke, P., Shinde, B., & Totala, N. (2014). Fabrication and analysis of counter flow helical coil heat exchanger. International Journal of Engineering Trends and Technology (IJETT)–Volume, 15(5), 229-240.
[8] Karthick, M., Kumar, S. R., Athiappan, K., & Gnanaraj, S. J. P. (2022, April). Thermal analysis of counter flow concentric tube heat exchanger using Runge-Kutta method. In AIP Conference Proceedings (Vol. 2405, No. 1, p. 050008). AIP Publishing LLC.
[9] Jia, R., Hu, J., & Elbalsohi, A. E. (2014, April). Analysis of a counter flow parallel-plate heat exchanger. In 2014 ASEE Zone 1 Conference.
[10] Subramanian, R., Senthil Kumar, A., Vinayagar, K., & Muthusamy, C. (2020). Experimental analyses on heat transfer performance of TiO2–water nanofluid in double-pipe counter-flow heat exchanger for various flow regimes. Journal of Thermal Analysis & Calorimetry, 140(2).
[11] Sridharan, M. (2022). Performance optimization of counter flow double pipe heat exchanger using grey relational analysis. International Journal of Ambient Energy, 43(1), 5318-5326.
[12] Dvořák, V., & Vít, T. (2015). Numerical investigation of counter flow plate heat exchanger. Energy Procedia, 83, 341-349.
[13] Zhan, C., Duan, Z., Zhao, X., Smith, S., Jin, H., & Riffat, S. (2011). Comparative study of the performance of the M-cycle counter-flow and cross-flow heat exchangers for indirect evaporative cooling–paving the path toward sustainable cooling of buildings. Energy, 36(12), 6790-6805.
[14] Sammeta, H., Ponnusamy, K., Majid, M. A., & Dheenathayalan, K. (2011). Effectiveness charts for counter flow corrugated plate heat exchanger. Simulation Modelling Practice and Theory, 19(2), 777-784.
[15] Cabezas-Gomez, L., Aparecido Navarro, H., & Maria Saiz-Jabardo, J. (2007). Thermal performance of multipass parallel and counter-cross-flow heat exchangers. Journal of heat transfer, 129(3), 282-290.
[16] Saeid, N. H., & Seetharamu, K. N. (2006). Finite element analysis for co-current and counter-current parallel flow three-fluid heat exchanger. International Journal of Numerical Methods for Heat & Fluid Flow, 16(3), 324-337.