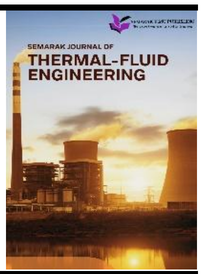
Semarak Journal of Thermal-Fluid Engineering

Journal homepage:
https://semarakilmu.my/index.php/sjotfe/index
ISSN: 3030-6639



Computational Modelling on Conjugate Heat Transfer in a Finned Pipe Unit

Muhammad Aizat Mohd. Nurudin[1], Ishkrizat Taib[1,*]

[1] Department of Mechanical Engineering, Faculty of Engineering Mechanical and Manufacturing, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:
Received 26 April 2026
Received in revised form 5 May 2026
Accepted 22 May 2026
Available online 30 June 2026

Keywords:
Conjugate; double-pipe; CFD

ABSTRACT

Double-pipe heat exchangers are fundamental thermal management devices that rely on conjugate heat transfer to exchange energy between fluids. While modern research extensively explores complex passive enhancements like finned surfaces to improve thermal efficiency, these modifications introduce significant pressure drops and obscure the fundamental physics of the heat exchange. Consequently, there is a critical need for high-fidelity, baseline numerical data that isolates the pure effects of varying flow regimes in unmodified pipes. This study aims to establish this baseline by systematically quantifying the isolated impact of fluid velocity on the conjugate heat transfer performance of a smooth double-pipe heat exchanger. A three-dimensional computational fluid dynamics (CFD) simulation was conducted to model the cooling of hot liquid ethylene using water as the coolant. A structured hexahedral mesh was utilized, achieving grid independence at 47,614 elements with a microscopic 0.005% outlet temperature variation after targeted refinement of the inner tube surface. The ethylene mass flow rate was systematically varied to analyze the thermal behavior across laminar, transitional, and turbulent flow regimes. The numerical analysis demonstrated that increasing the ethylene mass flow rate from 0.00178 kg/s (laminar) to 11.0 kg/s (turbulent) caused a transition from viscous-dominated to inertia-driven flow. Although this regime shift thinned the thermal boundary layer and enhanced the local convective heat transfer, the significantly reduced residence time of the fast-moving fluid ultimately dominated the overall thermal exchange. Consequently, the net cooling of the ethylene was reduced at higher velocities, with the outlet temperature increasing from 326.89 K in the laminar regime to 341.14 K in the turbulent regime. It is concluded that the balance between internal flow regimes and fluid residence time fundamentally dictates the efficiency of the conjugate heat transfer process. The validated simulation provides an accurate, highly reliable baseline of smooth-pipe thermal characteristics, which is essential for accurately evaluating the true net benefits of future geometric enhancements in industrial waste heat recovery systems.

[*] *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/sjotfe.9.1.1322a

1. Introduction

Heat exchangers are fundamental engineering devices built for efficient heat transfer from one fluid to another medium, playing a vital role in process industries and power plants, as described by Kumar *et al.,* [1]. Among the various designs available, double-pipe heat exchangers (DPHE's) are widely used in thermal systems due to their simplicity and range, and enhancing their operational performance is critical for overall energy efficiency as noted by Alwan *et al.,* [2]. The heat transfer mechanism in these concentric systems relies on the coupled interaction of fluid convection and solid conduction through the pipe wall, a process known as Conjugate Heat Transfer (CHT). Accurately predicting the thermal performance of such a system requires a detailed evaluation of the fluid flow behavior and the resulting temperature changes at the outlets. While extreme applications such as liquid metals in nuclear reactors often require highly complex four-equation turbulence models to capture CHT accurately as noted by Su *et al.,* [3], standard pressure-based CFD solvers are highly effective for evaluating conventional fluids like ethylene and water.

The internal flow dynamics of the heat exchanger are mathematically characterized by the Reynolds number, which represents the function of inertia and viscous forces as established by Ramkumar *et al.,* [4]. At low mass flow rates where the Reynolds number is low, viscous forces dominate, which typically leads to weaker convection and lower heat transfer rates [4]. Conversely, as the mass flow rate increases, the inertia force overwhelms the viscous forces, this thins the thermal boundary layer and enhances the convective heat transfer [4]. Understanding this relationship is essential for identifying the optimal balance between thermal performance and the thermodynamic irreversibilities often associated with different flow regimes [2].

Current studies often investigate passive heat transfer methods to overcome thermal resistance and optimise this balance. For example, Kumar *et al.,* [5] investigated the performance of a concentric tube heat exchanger with passive longitudinal fin profiles and found some thermal improvements for the rectangular and triangular shapes. Similarly, Seo *et al.,* [6] carried out experimental and numerical analysis to study the heat transfer characteristics of finned geometries and finally used Artificial Neural Networks (ANN) to predict the performance conditions.

Despite the extensive use of double-pipe heat exchangers (DPHEs) in industry and the development of advanced predictive models [5,6], there remains a need to establish clear numerical baselines that isolate the fundamental effects of varying Reynolds numbers on conjugate heat transfer without the interference of complex internal geometries such as fins. Quantifying this precise transition from viscous-dominated to inertia driven flow is highly significant for optimizing industrial energy efficiency and establishing reliable validation data for computational models.

Furthermore, the reliability of such numerical baselines depends fundamentally on the accuracy of the computational solvers used to capture these boundary interactions. As demonstrated in a recent numerical investigation of conjugate heat transfer in circular pipes by Doğan [7], accurately predicting thermal performance requires the simultaneous resolution of wall thickness, material thermal conductivity, and external convective forces. Doğan's findings underscore that utilizing high-fidelity CFD to isolate these foundational variables in smooth geometries is a critical prerequisite before introducing turbulent-inducing geometric enhancements.

Moving beyond fundamental fluid dynamics, these high-fidelity baselines play a very real role in designing and optimizing industrial waste heat recovery systems. As Zhai *et al.,* [8] point out, successfully recovering thermal energy in these settings hinges on maximizing the heat transfer performance of the specific fluids involved. In the real world, processing fluids like ethylene need to be cooled efficiently. However, if that cooling introduces severe pressure drops, it completely erases the thermodynamic and financial advantages of having a recovery system in the first place [9-12].

Ultimately, a deep numerical understanding of smooth-pipe thermal characteristics isn't just theoretical, it provides the essential benchmark needed to calculate the actual net benefits of these industrial applications [13-20].

Therefore, this study aims to simulate a three-dimensional pipe-in-pipe heat exchanger using Computational Fluid Dynamics (CFD). The overall heat transfer coefficient and effectiveness are numerically studied and calculated for a simple double pipe heat exchanger as proposed by Kumar [1]. Strictly following the geometry and boundary conditions of a standard concentric tube unit, the simulation investigates the cooling of hot liquid ethylene using water as the coolant. The study systematically varies the ethylene mass flow rate to observe behavior across the laminar, transitional, and turbulent regimes. This research will strictly analyze the resulting outlet temperatures of both the hot and cold fluids to determine how different flow conditions influence the efficiency of the conjugate heat transfer process.

2. Methodology

2.1 Geometry of a Three-Dimensional Concentric Double-Pipe Heat Exchanger

The computational domain consists of a three-dimensional concentric double-pipe heat exchanger. The model was constructed following standard concentric pipe dimensions to isolate the effect of flow dynamics on thermal transport. The overall length of the heat exchanger is 2.5 m. The outer shell has a diameter of 0.5 m with inlet and outlet diameters of 0.1 m as shown in Figure 1. The inner process pipe, carrying the hot fluid, maintains a uniform diameter of 0.1 m throughout. The fluid domains for both the hot ethylene and cold water were extracted to ensure accurate fluid-solid interfaces for the conjugate heat transfer simulation. The geometrical parameters of the model are summarized in Table 1.

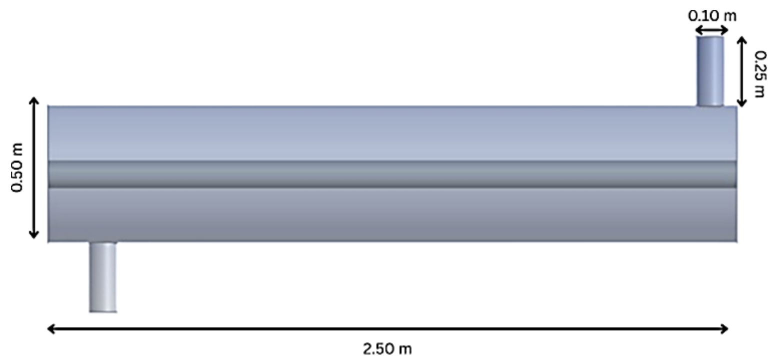


Fig. 1. Dimension of the concentric double-pipe heat exchanger

Table 1
Geometry parameters

Parameter	Value (m)
Overall pipe length	2.5
Outer shell diameter	0.5
Inlet and outlet pipe diameter (hot and cold)	0.1

2.2 Governing Equations

The fluid flow and heat transfer in the heat exchanger were solved using the conservation equations of mass, momentum, and energy. In this study, the governing equations were used to determine velocity, pressure, and temperature fields inside the heat exchanger. The equations of the continuity, momentum and energy shown as below.

$$\nabla . V^{\rightarrow} = 0 \tag{1}$$

$$\rho(V^{\rightarrow}.\Delta)V^{\rightarrow} = -\nabla p + \mu\nabla^2 V^{\rightarrow} \tag{2}$$

$$\rho C_p(V^{\rightarrow}.\nabla T) = k\nabla^2 T \tag{3}$$

where $V^{\rightarrow}$ is the velocity vector, ρ is fluid density, p is pressure, μ is dynamic viscosity, C_p is specific heat capacity, T temperature and k is thermal conductivity.

2.3 Numerical Setup and Material Properties

The 3D numerical simulation was conducted using ANSYS Fluent. As noted by Alwan *et al.,* [2], numerical simulations must be conducted under identical thermal and flow conditions to compare performance accurately. A pressure-based solver was employed with a steady-state formulation. For the turbulent flow case, the Realizable k-epsilon turbulence model was utilized.

To ensure the physical accuracy of the simulation, the material properties were strictly defined. The cooling water was modeled with a density of 998.2 kg/m³, a dynamic viscosity of 0.3 × 10^{-5} kg/m·s, a thermal conductivity of 0.6 W/m·K, and a specific heat of 4182 J/kg·K. The hot ethylene was assigned a density of 1.137 kg/m³, a dynamic viscosity of 1.03e-5 kg/m·s, a thermal conductivity of 0.0214 W/m·K, and its specific heat was defined using a piecewise polynomial function to capture temperature-dependent variations. The material properties are summarized in Table 2.

Table 2
Material properties

Material	Density (kg/m^3)	Viscosity (kg/m.s)	Thermal conductivity (W/m.K)	Specific heat (J/kg.K)
Water	998.2	0.3 × 10^{-5}	0.6	4182
Ethylene	1.137	1.03e-5	0.0214	Piecewise polynomial

2.4 Boundary Conditions and Flow Regimes

The boundary conditions were defined to evaluate the outlet temperatures effectively. The hot ethylene enters the inner pipe at a specified inlet temperature of 343 K, while the cooling water enters the shell at 283 K. The water mass flow rate was held constant at 6 kg/s for all simulations. Crucially, the interface between the fluids and the inner pipe was defined as a coupled wall to facilitate conjugate heat transfer between the solid wall and the fluids. This setup ensures that the overall heat transfer coefficient and effectiveness are numerically studied for a simple double pipe configuration [2]. Refer Figure 2 and Table 3 for more detail.

To study the differences of different flow dynamics, the mass flow rate of the ethylene was varied based on the Reynolds number (Re) which are 2200 for Laminar, 3000 for transitional and 1.36 × 10^7 for turbulence, calculated from Eq. (4) and summarize in Table 3. which represents the function of inertia and viscous forces [4]. The mass flow rates ($\dot{m}$) were specified as follows:

- Laminar flow (0.00178 kg/s): Where viscous forces dominate, resulting in weaker convection [4].

- Transitional flow (0.00243 kg/s): Representing the shift toward turbulence.
- Turbulent flow (11.0 kg/s): Where inertia becomes dominant, thinning the thermal boundary layer and enhancing convection [4].

$$Re = \frac{4\dot{m}}{\pi D \mu} \quad (4)$$

Rearrange the equation to find mass flow rate it becomes,

$$\dot{m} = \frac{\pi D Re \mu}{4} \quad (5)$$

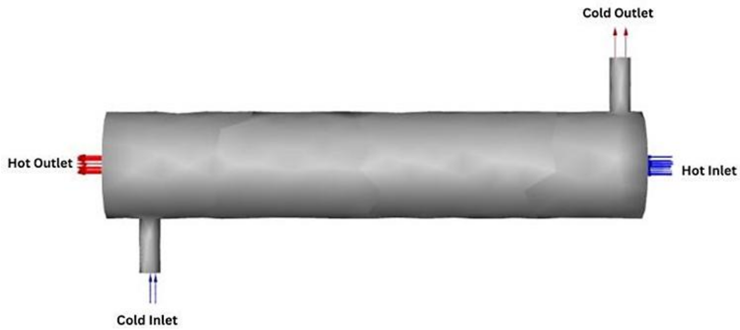


Fig. 2. Boundary Conditions setup for the concentric double-pipe heat exchanger

Table 3
Boundaries conditions

	Water	Ethylene
Inlet temperature, (K)	283	343
Reynold number (Re)	7.62×10^4	2.2×10^3, 3×10^3, 1.36×10^7
Type of flow	Turbulence	Laminar, transitional, turbulence
Mass flow rate, $\dot{m}$ (kg/s)	6	1.78×10^{-3}, 2.43×10^{-3}, 11.0

2.4 Grid Independence Test (GIT) and Meshing

To optimize computational efficiency and align the mesh with the axial flow direction, a structured hexahedral mesh was generated using the Sweep method. A Grid Independence Test (GIT) was performed on the turbulent case to ensure the accuracy of the outlet temperature readings. Specifically, the test systematically evaluated three progressively refined mesh configurations coarse, medium, and fine. To accurately resolve the critical convective boundary layer without unnecessarily inflating the global cell count, this progressive refinement was applied exclusively to the inner tube surface in direct contact with the ethylene fluid (Figure 3). A mesh convergence analysis was then conducted by monitoring the ethylene outlet temperature against the increasing number of elements.

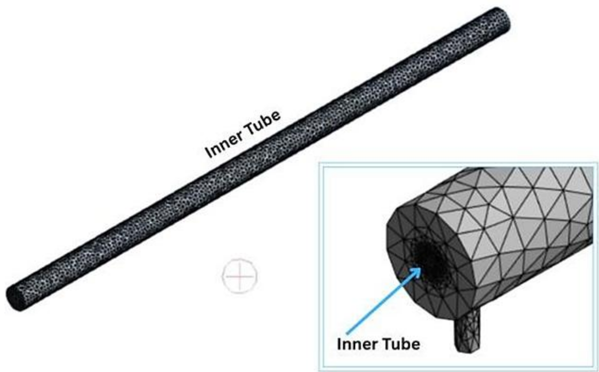


Fig. 3. Computational mesh of the concentric double-pipe heat exchanger

2.5 Solution Setup and Convergence

After the boundary conditions, material properties, and turbulence models fully defined, the model was solved under steady-state conditions. The energy equation remained activated throughout the process to accurately capture the conjugate heat transfer exchanging between the hot ethylene and the cold water. Before executing the calculation, the computational domain was initiaized using Hybrid Initialization, and the solver was configured to run for 500 iterations. Achieving strict convergence is critical to obtaining a mathematically sound and reliable CFD solution. In this study, convergence was evaluated by actively tracking the standard mathematical residuals. The calculation was considered fully converged, and final results were extracted, once these residuals dropped below acceptable error thresholds and stabilized, as depicted in the residual plot in Figure 4. This ensured that the governing equations had successfully reached a true steady-state balance.

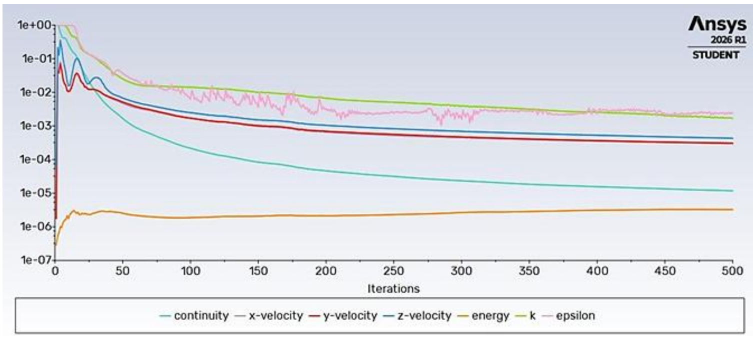


Fig. 4. Residual plot

3. Results

3.1 Grid Independence Test

Three mesh densities were tested, namely coarse, medium, and fine mesh. The hot outlet temperature and cold outlet temperature were selected as comparison parameters because they represent the main thermal of the concentric double-pipe heat exchanger. The Grid Independence Test results are presented in Figure 5 and Table 4. Analysis of the GIT data revealed that across the tested configurations 24,852, 47,614, and 142,505 elements, the outlet temperature stabilized rapidly. While the temperature shifted from 340.7855 K to 341.0423 K between the coarse and medium grids, refining the mesh further to 142,505 elements only yielded a microscopic fluctuation to 341.0592 K. This represents a relative variation of just 0.005% between the medium and fine grids, which is well below the standard 1% threshold required for CFD validation. Therefore, a grid-independent solution was achieved, and the medium mesh density of 47,614 elements was selected for all subsequent simulations to prevent numerical diffusion while conserving computational resources.

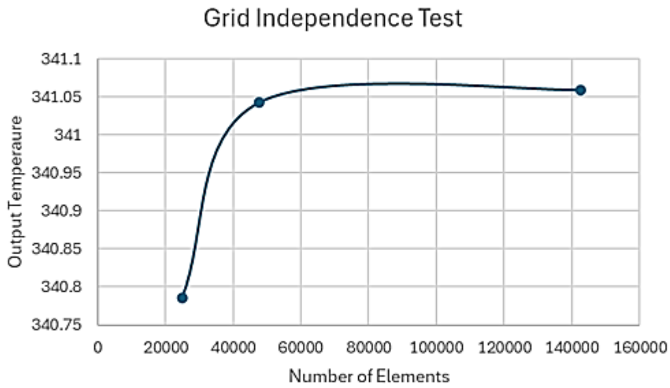


Fig. 5. Grid independence test

Table 4
Grid independence test result

Mesh type	Number of elements	Hot outlet temperature (K)	Cold outlet temperature (K)
Coarse	24852	340.7855	315.3074
Medium	47614	341.0423	315.3073
Fine	142505	341.0592	315.3074

3.2 Mass Flow Rate Comparison

The numerical simulation successfully evaluated the conjugate heat transfer across the double-pipe heat exchanger under identical thermal boundary conditions, fulfilling the performance assessment methodology outlined by Alwan *et al.,* [2]. The fixed inlet temperatures were 343 K for the hot ethylene and 283 K for the cooling water. Table 5 presents the final recorded outlet temperatures for both fluids across the three specified mass flow regimes.

Looking at the outlet temperatures in Table 5, there is a clear divide in cooling performance across the different flow regimes. Interestingly, the most substantial temperature drop actually happens during laminar flow, where the ethylene cools by 16.12 K dropping from an initial 343 K down to 326.88 K. As the flow shifts into the transitional regime, that outlet temperature ticks up slightly to 327.44 K.

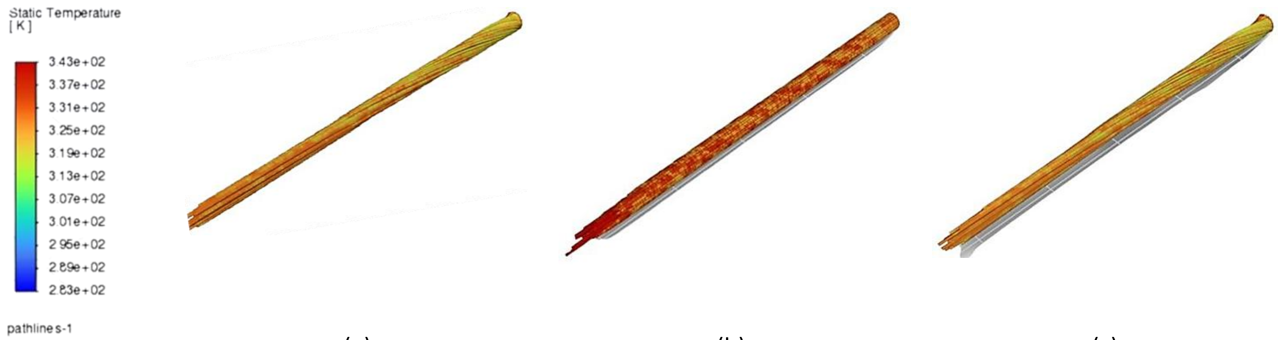
Table 5
Simulated outlet temperature across flow regimes

Flow regime	Ethylene mass flow rate (kg/s)	Ethylene		Water	
		Input (℃), K	Output (℃), K	Input (℃), K	Output (℃), K
Laminar	0.00178	343	326.88766	283	315.45655
Transitional	0.00243	343	327.44241	283	315.45647
Turbulence	11.00	343	341.1418	283	315.4564

The real shift, however, shows up in the turbulent regime. Even though turbulent flow naturally thins the boundary layer and boosts local convective heat transfer, the ethylene barely cools at all—dropping just 1.86 K to exit at 341.14 K. This inverse relationship between the mass flow rate and the absolute cooling really highlights that, for this specific smooth-pipe setup, residence time dictates everything. At 11.00 kg/s, the ethylene is simply moving too fast; it shoots through the heat exchanger before it has a chance to transfer its heat to the walls, completely wiping out the convective advantages you would normally expect from turbulence.

3.3 Temperature Distribution

The temperature contour comparison is shown in Figure 6. The contour was used to observe the temperature distribution inside the double-pipe heat exchanger for the three inlet velocity cases. Temperature contour plots are commonly used in heat exchanger CFD studies to visualize thermal behaviour and outlet temperature response.

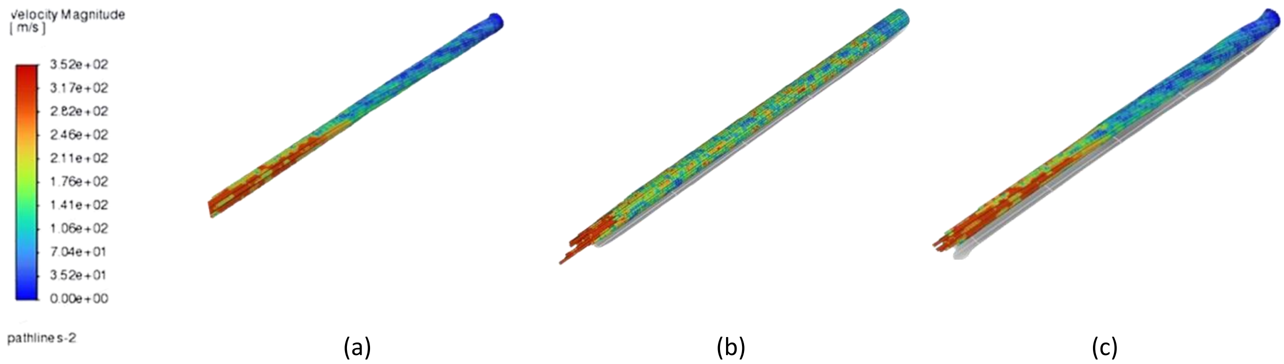


(a) (b) (c)

Fig. 6. Temperature path line for (a) Laminar (b) Transitional (c)Turbulence flow

3.4 Velocity Streamline Behaviour

Figure 7 shows the velocity streamlines across the different flow regimes. These plots offer a clear look at what is actually happening inside the pipe, making it easy to track the internal flow paths, the velocity distribution, and the boundary layer development down its length. In a smooth-pipe setup, streamlines are incredibly useful because they provide visual proof of the flow's transition shifting from a highly organized, viscous laminar profile straight into the fast, chaotic mixing of a turbulent state.



(a) (b) (c)

Fig. 6. Velocity path line for (a) Laminar (b) Transitional (c)Turbulence flow

4. Conclusions

In conclusion, this study successfully established a high-fidelity numerical baseline quantifying the isolated impact of fluid velocity on conjugate heat transfer in a smooth double-pipe heat exchanger. The validated CFD data demonstrates a critical operational trade-off dictated by the internal flow regime. While laminar flows at 0.00178 kg/s allow for a greater absolute temperature drop, cooling the ethylene by 16.11 K due to prolonged fluid residence time, the system's overall heat transfer rate is severely constrained by a thick, viscous thermal boundary layer. Conversely, turbulent flows at 11.0 kg/s overcome these viscous limitations. Although the drastically reduced residence time results in a much smaller temperature drop for the fluid itself cooling by only 1.86 K, the inertia-driven turbulence fundamentally thins the boundary layer and maximizes the convective heat transfer mechanism across the pipe wall [2,4]. Ultimately, this research confirms that fluid velocity and residence time fundamentally govern the efficiency of the conjugate heat transfer process, providing the essential smooth-pipe baseline required for evaluating future geometric enhancements in industrial systems.

References

[1] Kumar, Shiva, K. Vasudev Karanth, and Krishna Murthy. "Numerical study of heat transfer in a finned double pipe heat exchanger." *World Journal of Modelling and Simulation* 11, no. 1 (2015): 43-54.

[2] Alwan, Ali Arkan, Zena Kadhem, Hussain Al-Kayiem, and Shatha Ali Merdan. "Entropy Generation and Performance Assessment of Finned Tubes in Double-Pipe Heat Exchangers." *Wasit Journal of Engineering Sciences* 13, no. 3 (2025): 62-73. https://doi.org/10.31185/wjes.Vol13.Iss3.739

[3] Su, Xing-Kang, Xian-Wen Li, Xiang-Yang Wang, Qi-Jian Chen, Qian-Wan Shi, Jing Qiu, and Long Gu. "Numerical study on conjugate heat transfer characteristics of liquid lead-bismuth eutectic and supercritical carbon dioxide in a PCHE straight channel." *Annals of Nuclear Energy* 186 (2023): 109756. https://doi.org/10.1016/j.anucene.2023.109756

[4] Ramkumar, P., G. Thilagapathy, A. Thinagar, M. Chendur, B. Loga Rajakumari, and S. Mothibala. "Investigation of Finned heat pipe on compact heat exchanger in waste heat recovery application–Experimental analysis." In *EPJ Web of Conferences*, vol. 336, p. 01007. EDP Sciences, 2025. https://doi.org/10.1051/epjconf/202533601007

[5] Kumar, Shiva, K. Vasudev Karanth, and Krishna Murthy. "Numerical study of heat transfer in a finned double pipe heat exchanger." *World Journal of Modelling and Simulation* 11, no. 1 (2015): 43-54.

[6] Seo, Young Min, Ho Yeon Choi, Rock Kil Ko, Seokho Kim, and Yong Gap Park. "Evaluation of extrapolation ability of artificial neural network modeling on the heat transfer performance of a finned heat pipe." *Journal of Mechanical Science and Technology* 38, no. 12 (2024): 6657-6671. https://doi.org/10.1007/s12206-024-1122-9

[7] Dogan, Bekir. "Numerical investigation of conjugate heat transfer in circular pipes considering wall thickness, material conductivity, and external convection." *Results in Engineering* 29 (2026): 109236. https://doi.org/10.1016/j.rineng.2026.109236

[8] Zhai, Yu, Xu Zhao, Guanghui Xue, and Zhifeng Dong. "Study on heat transfer performance and parameter improvement of gravity-assisted heat pipe heat transfer unit for waste heat recovery from mine return air." *Energies* 16, no. 17 (2023): 6148. https://doi.org/10.3390/en16176148

[9] Ashraf, Ghazala, S. Bilal, Muhammad Ishaq, Khalid Saifullah Syed, A. S. Alqahtani, and M. Y. Malik. "Energy transmission through conjugate heat transfer in arrow-finned double exchanger by executing FEM simulations." *Case Studies in Thermal Engineering* (2025): 106855. https://doi.org/10.1016/j.csite.2025.106855

[10] Drag, Tomasz, and Mieszko Tokarski. "Heat transfer optimization via finned surface with the use of CFD." In *Journal of Physics: Conference Series*, vol. 2812, no. 1, p. 012017. IOP Publishing, 2024. https://doi.org/10.1088/1742-6596/2812/1/012017

[11] Hidman, Niklas, Daniel Almgren, Kim Johansson, and Eskil Nilsson. "How internal fins enhance the thermohydraulic performance of geothermal pipes: A direct numerical simulation study." *International Journal of Heat and Mass Transfer* 256 (2026): 128114. https://doi.org/10.1016/j.ijheatmasstransfer.2025.128114

[12] Hosseinkhani, Abolfazl, Sina Khaleghi, Seyyed Abdolreza Gandjalikhan Nassab, and Mohammad Hadi Mohammadi. "Exhaust gas heat recovery using a double-pipe heat exchanger with helical fins, considering gas radiation effect; numerical investigation, performance analysis, and optimization." *Energy Conversion and Management* 339 (2025): 119961. https://doi.org/10.1016/j.enconman.2025.119961

[13] Iqbal, Zafar, K. S. Syed, and M. Ishaq. "Fin design for conjugate heat transfer optimization in double pipe." *International Journal of Thermal Sciences* 94 (2015): 242-258. https://doi.org/10.1016/j.ijthermalsci.2015.03.011
[14] Kalantari, Hosein, Seyed Ali Ghoreishi-Madiseh, Jundika C. Kurnia, and Agus P. Sasmito. "An analytical correlation for conjugate heat transfer in fin and tube heat exchangers." *International Journal of Thermal Sciences* 164 (2021): 106915. https://doi.org/10.1016/j.ijthermalsci.2021.106915
[15] Lai, Xi, Jianghan Chen, Dier Ma, Ziying Luo, Shiran Bao, Limin Qiu, Kai Wang, and Xiaoqin Zhi. "Numerical simulation of conjugate heat transfer in pre-cooling process of cryogenic propellant transfer line by cryocoolers." *International Journal of Refrigeration* (2026). https://doi.org/10.1016/j.ijrefrig.2026.02.027
[16] Mansour, J., Zena Khalefa Kadhim, and Kamil Abdul Hussein. "CFD study of heat transfer characteristics for annular serrated finned-tube heat exchanger." *Journal of Computer and Engineering Technology* 5, no. 1 (2018): 77-87.
[17] Deguchi, Y., R. Muranaka, T. Kamimoto, T. Takagi, S. Kikuchi, and A. Kurihara. "Reaction path and product analysis of sodium-water chemical reactions using laser diagnostics." *Applied Thermal Engineering* 114 (2017): 1319-1324. https://doi.org/10.1016/j.applthermaleng.2016.09.167
[18] Rao, G. Arvind, and Yeshyahou Levy. "A semi empirical methodology for performance estimation of a double pipe finned heat exchanger." In *Engineering Systems Design and Analysis*, vol. 48388, pp. 167-177. 2008. https://doi.org/10.1115/ESDA2008-59124
[19] Tohidi, Amir, Mohsen Ghadyani, and Mehrdad Azadeh. "Numerical Study of Heat Transfer Characteristics of Nanofluid Flow in the Finned Pipe Subjected to Variable Heat Flux." *Res. Appl. Therm. Eng.* 3 (2020): 1-14.
[20] Doğan, Bekir, and Ünsal Aybek. "Numerical investigation of Reynolds number effects on laminar conjugate heat transfer in a counter-flow double-pipe heat exchanger." In *6th International Paris Congress on Applied Sciences* (2026): 297-304.