"Investigating the heat transfer in double pipe heat exchanger through variation in holes on fins using CFD analysis"

OPEN ACCESS

AG-2023-1001

Volume: 3

Issue: 1

Month: March

Year: 2024

ISSN: 2583-7117

Published: 30.03.2024

Citation:

Satyam Kumar[1], Shivendra Singh[2], Dr. B. Suresh[3]. "Investigating the heat transfer in double pipe heat exchanger through variation in holes on fins using CFD analysis." International Journal of Innovations In Science Engineering And Management, vol. 3, no. 1, 2024, pp. 20–28.

Satyam Kumar[1] Shivendra Singh[2], Dr. B. Suresh[3]

[1]*Research Scholar, Department of Mechanical Engineering, Corporate Institute Of Science & Technology, Bhopal*
[2]*Asst Prof., Department of Mechanical Engineering, Corporate Institute Of Science & Technology, Bhopal*
[3]*Prof. & HOD, Department of Mechanical Engineering, Corporate Institute Of Science & Technology, Bhopal*

Abstract
In order to lower the temperature of the exhaust gases and lessen the impact of greenhouse gases, a "double pipe heat exchanger" was used in this research. Three new non-linear fin configurations on the heat exchanger body, a hybrid of two nanofluid types (Ag and Fe3O4), and nanoparticles with a volume concentration of 2.5% each are use in this research. For investigating used the K-epsilon (κ − ε) turbulence modeling to assess the embedded simulation in the commercial program ANSYS-FLUENT. According to the study's findings, increasing the fins' hole sizes reduces the exhaust gases' temperature by an average of 1 kelvin. A decrease of 7.74% in exhaust gas and a decrease of 67% in nano-fluid pressure. Thus, it can be concluded that the amount of fin holes has a more significant impact on enhancing the heat transfer rate compared to the hybrid nanoparticles. The study's findings point to the kind of nanofluid, as well as the characteristics of the fins' number, shape, and arrangement, as elements that effectively lower the temperature of diesel engine exhaust gasses.

Keyword: Exhaust gas, Nanoparticles, Heat transfer, Fins, Turbulence, etc.

I. INTRODUCTION

A heat exchanger is a device that allows heat to be transferred from one fluid to another or from one fluid to a solid surface in the absence of physical contact between the two. The term "fluid" is broad enough to include not just liquids but also gases and vapours. This means that heat-exchanging devices may function in a wide variety of ways depending on the flow arrangement. Whether heating or cooling, a heat exchanger allows the fluids to transmit their thermal energy without coming into direct touch with one another.[1] Power plants, HVAC systems, chemical processing facilities, vehicles, and commonplace home appliances like water heaters are just a few examples of the many sectors and uses that rely on heat exchangers for effective heat transmission. Although heat exchangers may take several forms, they always work on the same principle: moving heat from a hot source to a cooler one. The most common components of a heat exchanger are plates and tubes.[2] A huge surface area is what tubes and plates are used for when it comes to heat transmission. The two fluids move in opposite directions via these channels. Fluids may move in either a parallel or counterflow pattern, depending on the design and needs. To provide the most effective transmission of heat, the heat exchanger makes full use of the fluids' thermal contact.[3], [4]

One fluid may transfer its heat to another, which is colder, using a heat exchanger, which prevents the two fluids from mixing or coming into physical touch. Take a pipe with another pipe around it as an example. It is possible for a hot fluid to flow through the inner pipe and a colder fluid to flow through the outer pipe at the same time.

Since the warmer fluid would be able to raise the temperature of the cooler one at the same time, the two fluids may actually work together to lower their temperatures.[5], [6]

In the domain of “heat transfer enhancement” in “double pipe heat exchanger”s with counter-flow arrangement, a significant research gap exists concerning the exploration of advanced surface modifications and geometrical configurations tailored specifically for this configuration.[7]–[9] While conventional methods such as extended surfaces and turbulators have been extensively investigated, there remains a lack of studies focusing on innovative approaches that can substantially augment heat transfer rates while mitigating pressure drop penalties. Moreover, there is limited understanding of the synergistic effects between different enhancement techniques within the context of counter-flow heat exchangers, necessitating further exploration to unlock their full potential.[10], [11] Additionally, the influence of operational parameters such as fluid flow rates, temperature differentials, and Reynolds numbers on the efficacy of “heat transfer enhancement” mechanisms in counter-flow “double pipe heat exchangers” warrants deeper investigation. Furthermore, there is a lack of research addressing the practical challenges associated with implementing enhanced designs at scale, including manufacturability, cost-effectiveness, and compatibility with existing infrastructure. Addressing these research gaps can pave the way for the development of more efficient and sustainable heat transfer solutions tailored specifically for counter-flow double pipe heat exchangers, with implications spanning diverse applications ranging from industrial processes to renewable energy systems.[12]

II. OBJECTIVE

- To study the effect of Nano-fluid in waste heat recovery system.
- To study the effect of fins in Nano-fluid and exhaust air of waste heat recovery system.
- To study the effect in heat transfer due to circular holes in fins.
- To study the effect in variation in position and number of holes in fins.

III. RESEARCH AND METHODOLOGY

In the complete study of the double pipe heat exchanger, several procedural steps are followed to ensure a systematic and thorough analysis. Initially, the computational domain is designed using CATIA, which involves creating the necessary geometry for the heat exchanger. Once the design is finalized, it is saved and imported into ANSYS design modeler for further processing. In ANSYS design modeler, the imported design undergoes mesh generation, where the mesh elements are created to discretize the domain for computational fluid dynamics (CFD) analysis. During this process, names are assigned to different regions of the domain, which are crucial for setting up boundary conditions later in the CFD analysis.

After mesh generation, the next step involves treating the fluid and solid components with the appropriate materials. This ensures that the physical properties of the materials are accurately represented in the simulation. Once the materials are applied, boundary conditions are set up to define the flow behavior, heat transfer characteristics, and any other relevant parameters for the analysis. Subsequently, the computational model is set up in ANSYS Workbench, specifically using the Fluent module for fluid dynamics analysis. The Fluent model allows for the assessment of various performance metrics, such as heat transfer rates, pressure drops, and flow patterns, providing valuable insights into the behavior of the double pipe heat exchanger under different operating conditions. This comprehensive approach from design to analysis ensures a thorough understanding of the heat exchanger's performance and helps in optimizing its design for efficiency and effectiveness.

A. *Hybrid Nano-fluid*

The majority of the current equations that describe the characteristics of mixtures containing a certain kind of nanoparticle are highly congruent with the relevant experimental data. Nevertheless, there is a dearth of empirical correlations in the global literature that may be used to evaluate the thermos-physical characteristics of mixtures including hybrid nanoparticles. Preparation, stability, and characterization of hybrid nanoparticles have been the primary areas of research. Since it is assumed that the nanoparticles and PCMs are in thermal equilibrium, the density and specific heat capacity are determined using the formula shown below, by extending the mixing rule and stating that nanoparticles 1 while nanoparticles 2 are considered:

$$\rho_{comp} = \emptyset_1 \rho_{n1} + \emptyset_2 \rho_{n2} + (1 - \emptyset_1 - \emptyset_2)\rho_{PCM}$$

$$Cp_{comp} = \frac{\emptyset_1 \rho_{n1} Cp_1 + \emptyset_2 \rho_{n2} Cp_2 + (1 - \emptyset_1 - \emptyset_2)\rho_{PCM} Cp_{PCM}}{\rho_{comp}}$$

$$k_{comp} = k_{PCM}\frac{\left(\frac{\emptyset_1 k_{n1} + \emptyset_2 k_{n2}}{\emptyset_1 + \emptyset_2}\right) + 2k_{PCM} + 2(\emptyset_1 k_{n1} + \emptyset_2 k_{n2} - (\emptyset_1 + \emptyset_2)k_{PCM})}{\left(\frac{\emptyset_1 k_{n1} + \emptyset_2 k_{n2}}{\emptyset_1 + \emptyset_2}\right) + 2k_{PCM} - (\emptyset_1 k_{n1} + \emptyset_2 k_{n2} - (\emptyset_1 + \emptyset_2)k_{PCM})}$$

“Where, ρ, C_p, $\emptyset$, and k_n are the density, specific heat, volume concentration of nano particles and thermal conductivity of nano particles respectively. In this study two nanoparticles are use which is Ag and Fe_3O_4 and both having same volume concentration of 2.5 %. The thermos-physical properties are mention in table 3.3.”

B. Computational domain

The whole dimensions are in mm and mention in below figure. In this study, fin height is 24 mm and pattern of fin is illustrate through $y=20 \times \sin(20 \times \pi)$. Exhaust gas and hybrid Nano-fluid flow through inner tube and outer tube respectively. Nano fluid and Gas flow in opposite direction with each other.

In this case1 fin is smooth with height of 24 mm. In this case 2, all design are same which is mention in upper section except fin. There is 11 holes of 10 mm diameter present in all six fin. Center to center distance between 1st hole and last hole is 700 mm, and pitch is 70 mm. In this case 3, all design are same which is mention in upper section except fin. There is 13 holes of 10 mm diameter present in all six fin. Center to center distance between 1st hole and last hole is 720 mm, and pitch is 60 mm.

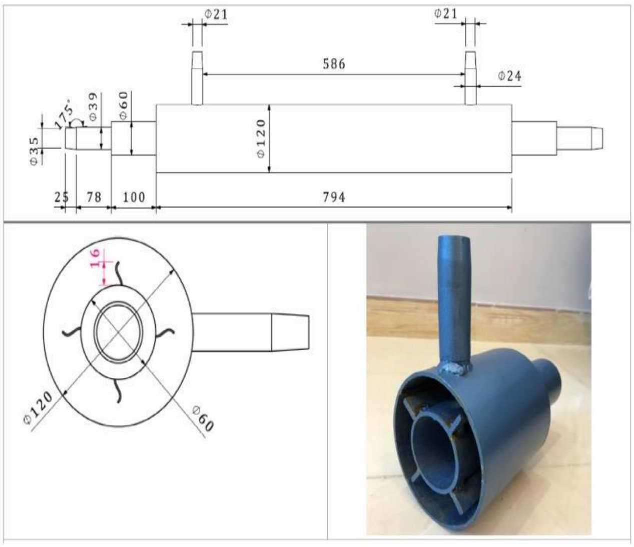
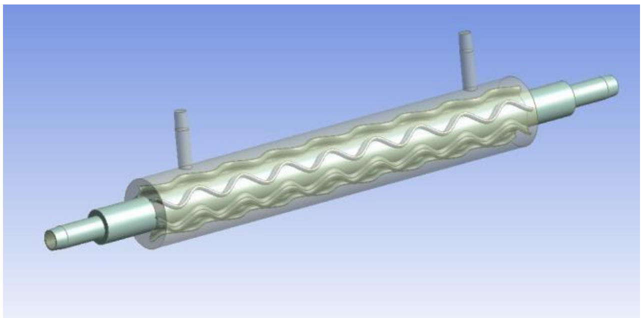
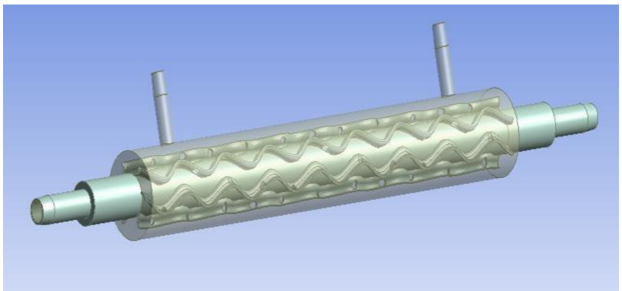


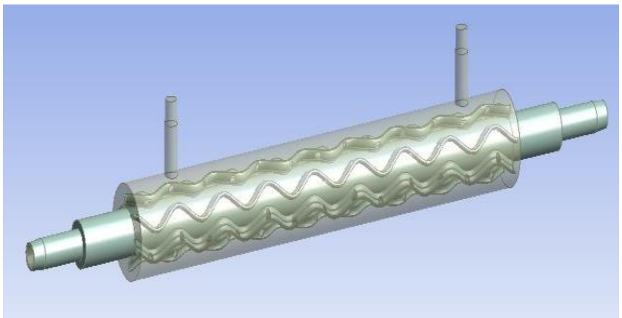
Figure 1 Dimension of computational domain



Case 1



Case 2



Case 3

Figure 2 Computational Domain

C. Meshing

In ANSYS, mesh generation refers to the process of creating a mesh, which is a discretized representation of the computational domain. This mesh consists of elements such as nodes, elements, and boundaries that allow for the numerical simulation of physical phenomena, such as fluid flow, heat transfer, and structural analysis. Mesh generation plays a crucial role in accurately representing the geometry, capturing the behavior of the system, and ensuring reliable simulation results. Shape of the element has tetrahedral selected. Number of element and nodes are mention in table below.

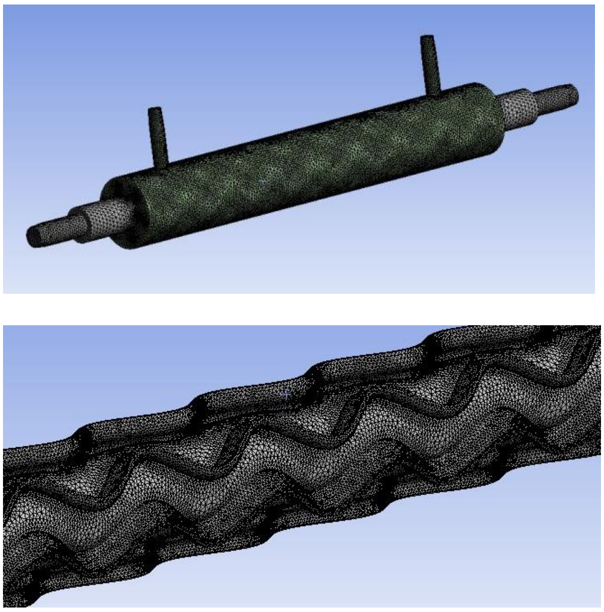


Figure 3 Mesh generation of computational Domain

Table 1 Number of Element and Nodes

	Element	Nodes
case 1	2682607	572828
case 2	3154807	663689
case 3	3229923	677910

D. Boundary condition

When solving the momentum equations using Fluent, the problem convergence criteria was set to 10-03. For the energy equation, the value was set to 10-04. Further, the turbulence kinetic energy and pressure equations were detected using the first- and second-order upwind approaches, respectively. In this model, the connection between pressure and velocity was also evaluated using the SIMPLE approach. A second-order upwind approach was used to discretize the convective components in the momentum and energy equations. The boundary conditions were considered to be the mass flow rate at the inputs and the pressure at the outputs. The walls were also evaluated for their no-slip properties. Additionally, it was assumed that the heat exchanger shell's outside wall was insulated, and the reverse heat wall for the interior walls was taken into account as the boundary condition for heat transfer. In Table 3 we can see the boundary condition and problem input values, and in Table 2 illustrate the thermodynamic and thermal properties of the heat exchanger's fluid.

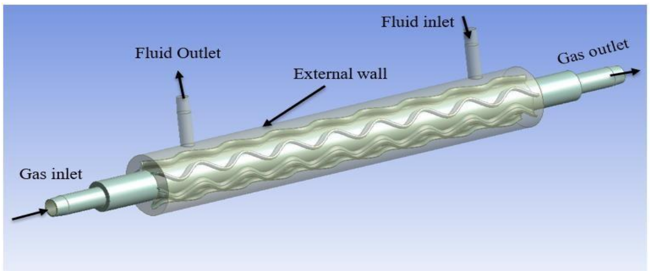


Figure 4 Boundary condition illustrate

Table 2 Thermo-physical properties of fluid and Nano-fluid

Physical Characteristics	Unit	Water	Ag	Fe_3O_4	Hybrid nanoparticle (2.5% - Ag + 2.5% Fe3O4 + 95% water)
Density	Kg/m3	997.1	10,500	5,200	1339.745
Specific Heat	J/kgK	4,179	235	670	3020.078
Thermal conductivity	W/mK	0.613	429	6	0.709
Viscosity	Kg/m.s				0.00114

Table 3 CFD boundary condition for Different Component

Domain	Boundary condition	Amount	Description
Inlet (Exhaust gases)	Mass flow inlet Temperature	0.00934 kg/s 508 K	Hydraulic diameter = 0.06 Intensity of turbulence = 3%
Inlet fluid	Mass flow inlet Temperature	0.0054 kg/s 305 K	Hydraulic diameter = 0.06 Intensity of turbulence = 3%

Outlet	Pressure outlet	0 Pa	Intensity of turbulence = 3%
Interior walls	Standard wall	Coupled	-
External wall	Standard wall	0 (heat flux)	Insulation
Turbulence model	(K – ε)	Standard	-

E. *Validation*

"In order to verify the correctness of the CFD findings, validation tests are run to examine the precision and detect any discrepancies in the numerical simulations. In order to verify the current simulation's outcome with (Gholizadeh et al., 2022) [13] the outcome of the simulation, compare the exhaust gas outlet in a shell and tube heat exchanger with smooth fins. The mass flow rate in the inner tube (Exhaust gas) is 0.00934 kg/s, whereas in the outer shell (Nano fluid) it is 0.0054 kg/s. Exhaust gas is 508 kelvin and Nanofluid is 305 kelvin at the starting stage of the simulation. Figure shows the verified results.

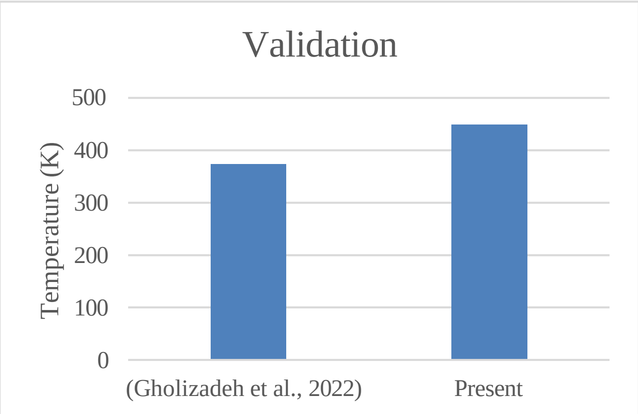
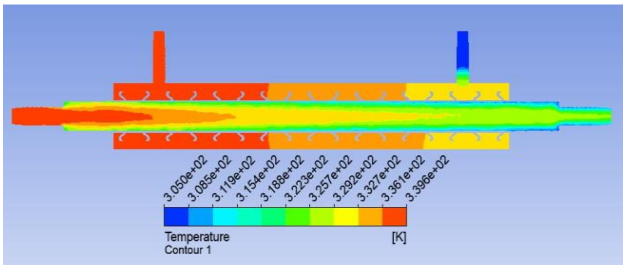


Figure 5 Validation at Exhaust outlet Temperature

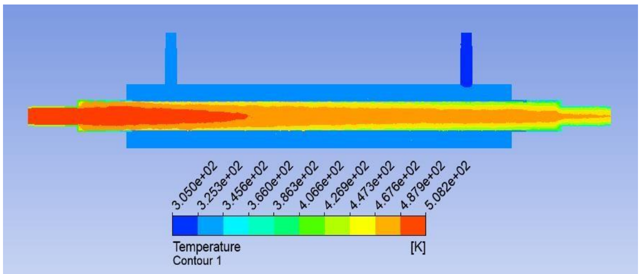
IV. RESULT AND DISCUSSION

A. *Temperature Contour*

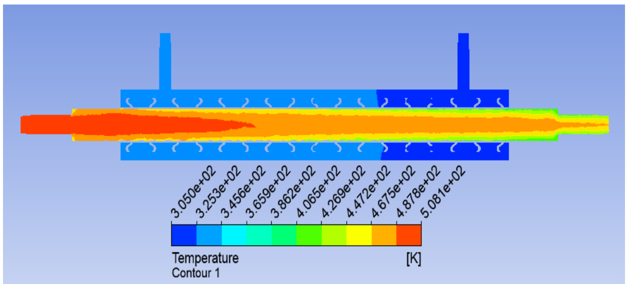
In the double pipe heat exchanger, the presence of fins strategically placed on the outer surface of the tube significantly influences the temperature distribution along the tube length. The fins, which vary in configuration with some having smooth surfaces and others containing 11 or 13 holes, serve as obstacles that alter flow dynamics and enhance heat transfer between the hot exhaust gas in the inner tube and the cold Nano-fluid in the outer tube or shell. As the exhaust gas flows through the inner tube, it undergoes a cooling process due to heat exchange with the Nano-fluid in the outer tube. Initially, the temperature distribution is mainly affected near the tube wall and around the fins. However, as the gas encounters each fin along the flow path, there is a noticeable change in temperature distribution that extends beyond the vicinity of the tube wall and fins.



Case 1



Case 2

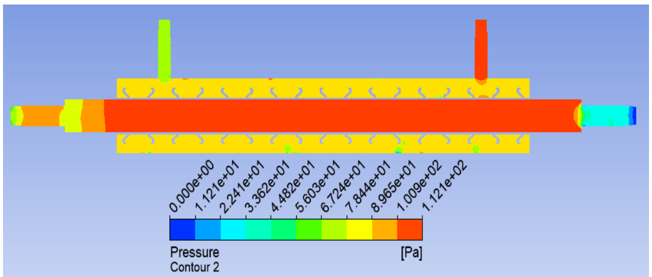


Case 3

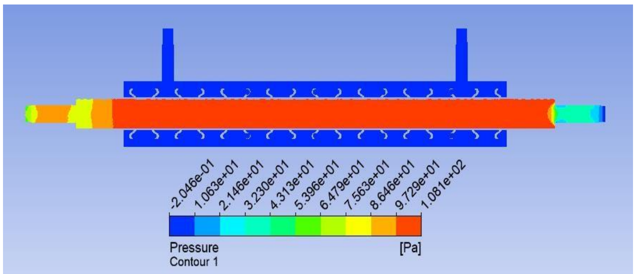
Figure 5 Temperature contour

B. *Pressure Contour*

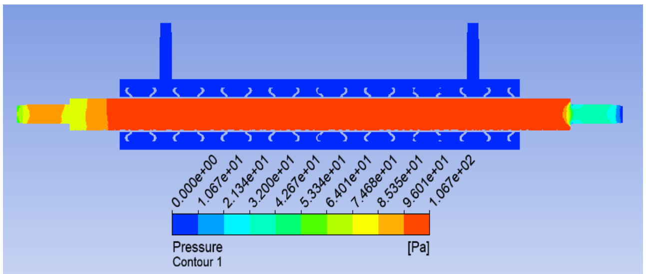
Conversely, in the outer tube or shell, where the nano fluid flows, the pressure distribution follows a similar trend but in the opposite direction. At the inlet of the shell, where the nano fluid enters, the pressure are high. As the nano fluid flows along the length of the shell, it undergoes a heating process due to the heat exchange with the exhaust gas in the inner tube. This increase in temperature is accompanied by a corresponding decrease in pressure. The pressure gradually decreases as the nano fluid progresses along the shell, reaching its lowest point at the outlet where it exits the shell. Similar to the inner tube, at the outlet of the shell, the pressure approaches zero. The presence of fins within the outer surface of the tube affects the pressure distribution by altering the flow dynamics. These smooth fins and holes on fins can induce turbulence and vortices in the flow, which may lead to localized fluctuations in pressure along the tube. However, overall, the pressure trend follows the general pattern described above, with pressure decreasing along the length of both the inner tube and the outer shell as show in the Figure 7.



Case 1



Case 2

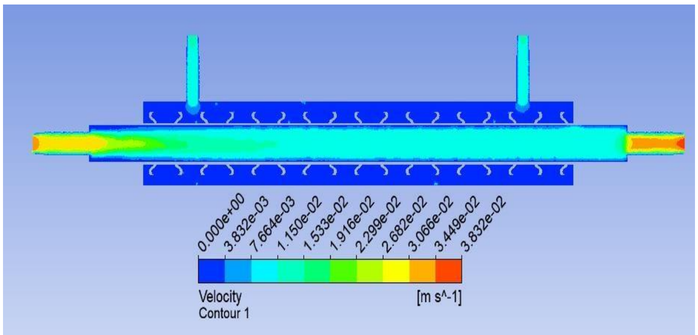


Case 3

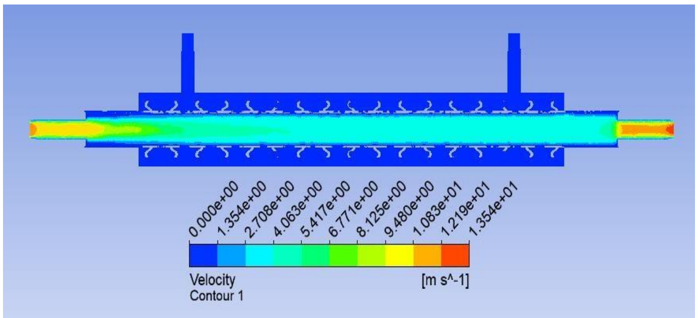
Figure 6 Pressure contour

C. Velocity Contour

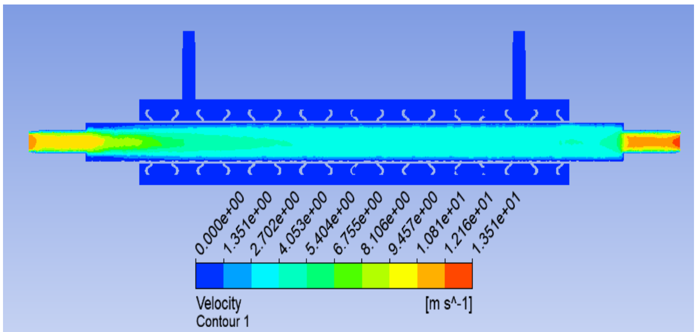
In the "double pipe heat exchanger", the dynamics of fluid flow within the inner tube, carrying the exhaust gas, and the outer tube or shell, containing the nano fluid, are crucial for efficient heat transfer. Within the inner tube, as the exhaust gas flow with in small diameter so at the inlet velocity is maximum. In contrast, within the outer tube or shell, the nano fluid enters at the inlet with relatively low velocity. As it progresses along the length of the shell, the velocity of the nano fluid increases. This acceleration in velocity occurs primarily due to presence of holes in fins. At the outlet of the shell, where the pressure approaches zero, the velocity of the nano water reaches its peak. This phenomenon is a result of Bernoulli's principle, which states that in a streamline flow, an increase in fluid velocity is accompanied by a decrease in pressure. Therefore, the velocity increase in the nano fluid within the outer tube is a consequence of the pressure drop along its length, ultimately leading to higher flow velocities at the outlet.



Case 1



Case 2



Case 3

Figure 7 Velocity contour

D. Comparison of results

Overall, the presence of holes in fins within the double pipe heat exchanger significantly improves its heat transfer by enhancing the convective heat transfer process between the exhaust gas flowing in the inner tube and the nano fluid circulating in the outer tube of the shell. Figure 11 illustrate that outlet temperature of the exhaust gas is slightly decreases and there is a major difference in case 2 and case 3 from the case 1. Figure 9 illustrate that the outlet temperature of nano fluid in case 3 is maximum. Temperature difference of exhaust gas and Nano fluid is maximum in case 3. In case of pressure difference in exhaust gas case 3 having a minimum value and Nano-fluid, case 3 having a minimum value illustrate in figure 10.

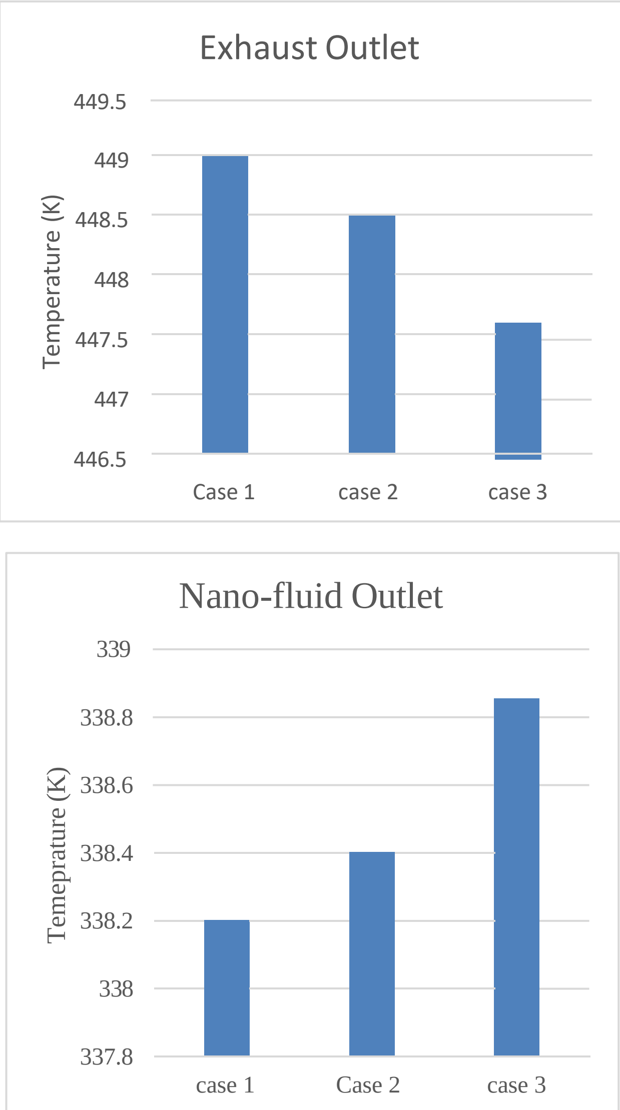


Figure 8 Exhaust gas and Nano-fluid outlet temperature of all cases

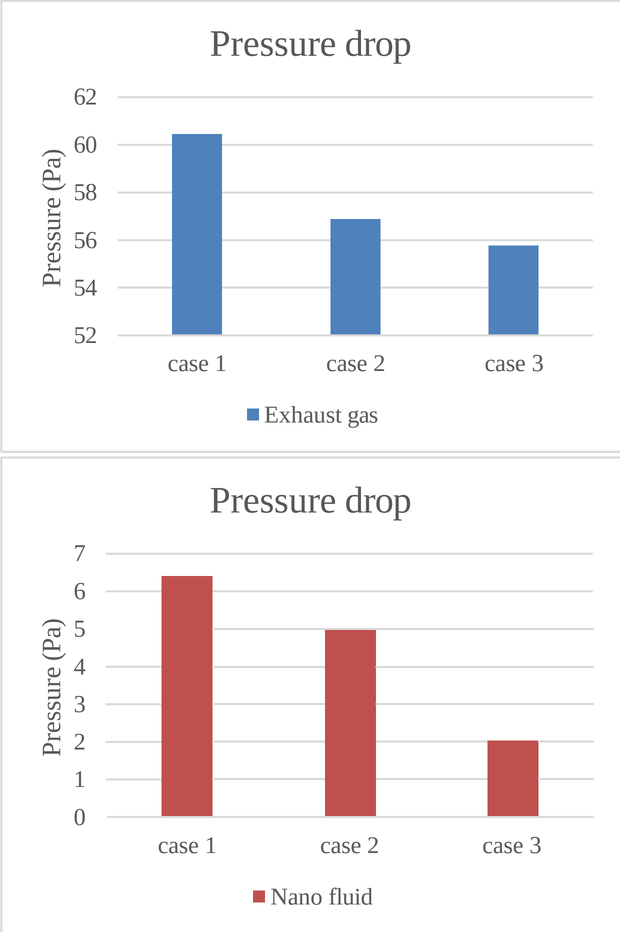


Figure 9 Pressure drop of exhaust gas and Nano-fluid

V. CONCLUSION

"In this study, the impact of using holes in fins in a double pipe Heat Exchanger numerically investigated. The simulation analysis held on three different cases of fin design in different locations to study the spatial impact of the fins on the enhancement of heat transfer. Thus, the heat transfer in each case has affected by geometry, which is the location of the holes in fins. In this study, fins located at outer surface of the inner tube. In general, results show that holes in fins enhance heat transfer in all the cases as shown by the outlet temperature, temperature difference and pressure at inner tube and outer tube. Results show that the turbulence flow occurred in both parts tube and annular with higher turbulence in the hot region, which is effective to enhance heat transfer. In more detail, the following points were concluded."

- Case 1 having a maximum pressure drop in both inner tube and outer tube, which is 60.44 and 6.4 Pa respectively.

- Case 3 having a minimum pressure drop in both inner tube and outer tube, which is 55.76 and 2.086 Pa respectively.
- In case outlet temperature of the nano fluid, case 1 having minimum value, which is 338.217 K and case 3 having maximum value, which is 338.855 K.
- In case of exhaust gas outlet temperature, case 1 having maximum value which is 448.993 K, and case 3 having minimum value which is 447.645 K.
- In the comparison of temperature drop, case 3 having a good result.

VI. FUTURE SCOPE

The future scope for enhancing "heat transfer in double pipe heat exchangers" with counter-flow arrangement is promising, with several avenues for exploration and innovation. Advanced computational modeling techniques, such as computational fluid dynamics (CFD) coupled with optimization algorithms, offer great potential for designing optimized geometries and surface modifications to increase "heat transfer efficiency" with the lessening of pressure loss. Furthermore, the integration of emerging materials with superior thermal conductivity and surface characteristics, such as nanomaterials and advanced alloys, holds promise for enhancing heat transfer rates in "counter-flow double pipe heat exchangers". Moreover, the application of additive manufacturing techniques opens up possibilities for the fabrication of complex geometries and customized heat transfer surfaces, facilitating the implementation of tailored enhancement strategies. Collaborative interdisciplinary research efforts involving experts from fields such as fluid dynamics, materials science, and control engineering will be essential for pushing the boundaries of heat transfer enhancement in counter-flow double pipe heat exchangers and unlocking their full potential across a wide range of industrial and environmental applications.

Reference

[1] L. Liu, Z. Cao, T. Shen, L. Zhang, and L. Zhang, "Experimental and numerical investigation on flow and heat transfer characteristics of a multi-waves internally spiral finned tube," *Int. J. Heat Mass Transf.*, vol. 172, 2021, doi: 10.1016/j.ijheatmasstransfer.2021.121104.

[2] F. A. Hamedani, S. S. M. Ajarostaghi, and S. A. Hosseini, "Numerical evaluation of the effect of geometrical and operational parameters on thermal performance of nanofluid flow in convergent–divergent tube," *J. Therm. Anal. Calorim.*, vol. 140, no. 3, pp. 1483–1505, 2020, doi: 10.1007/s10973-019-08765-w.

[3] F. H. Sani, M. Pourfallah, and M. Gholinia, "The effect of MoS2–Ag/ H2O hybrid nanofluid on improving the performance of a solar collector by placing wavy strips in the absorber tube," *Case Stud. Therm. Eng.*, vol. 30, no. January, p. 101760, 2022, doi: 10.1016/j.csite.2022.101760.

[4] M. Gholinia, A. A. Ranjbar, M. Javidan, and A. A. Hosseinpour, "Employing a new micro-spray model and (MWCNTs - SWCNTs) - H2O nanofluid on Si-IGBT power module for energy storage: A numerical simulation," *Energy Reports*, vol. 7, pp. 6844–6853, 2021, doi: 10.1016/j.egyr.2021.10.047.

[5] S. K. Gupta, H. Verma, and N. Yadav, "A review on recent development of nanofluid utilization in shell & tube heat exchanger for saving of energy," *Mater. Today Proc.*, vol. 54, no. October, pp. 579–589, 2022, doi: 10.1016/j.matpr.2021.09.455.

[6] P. Jalili, K. Kazerani, B. Jalili, and D. D. Ganji, "Investigation of thermal analysis and pressure drop in non-continuous helical baffle with different helix angles and hybrid nano-particles," *Case Stud. Therm. Eng.*, vol. 36, no. June, p. 102209, 2022, doi: 10.1016/j.csite.2022.102209.

[7] N. Sakib, "Design Construction And Performance Test Of A Cross Flow Heat Exchanger Supervised By : Dr . Nawsher Ali Moral Department of Mechanical Engineering Khulna University of Engineering and Technology Prepared By : Md . Fazla Rabbi Khulna , Bangladesh Departme," no. August 2013, 2020, doi: 10.13140/RG.2.2.16489.52326.

[8] B. Jalili, S. Sadighi, P. Jalili, and D. D. Ganji, "Characteristics of ferrofluid flow over a stretching sheet with suction and injection," *Case Stud. Therm. Eng.*, vol. 14, no. April, 2019, doi: 10.1016/j.csite.2019.100470.

[9] A. H. Ghobadi, M. Armin, S. G. Hassankolaei, and M. Gholinia Hassankolaei, "A new thermal conductivity model of CNTs/C2H6O2–H2O hybrid base nanoliquid between two stretchable rotating discs with Joule heating," *Int. J. Ambient Energy*, vol. 43, no. 1, pp. 3310–3321, 2022, doi: 10.1080/01430750.2020.1824942.

[10] A. H. Ghobadi and M. G. Hassankolaei, "Numerical treatment of magneto Carreau nanofluid over a stretching sheet considering Joule heating impact and nonlinear thermal ray," *Heat Transf. - Asian Res.*, vol. 48, no. 8, pp. 4133–4151, 2019, doi: 10.1002/htj.21585.

[11] K. Sridhar and K. Bicha, "Comparative Analysis of Parallel and Counter Flow Heat Exchangers," *Sci. Eng. Technol. Res.*, vol. 06, no. 04, pp. 0638–0644, 2017, [Online]. Available: www.ijsetr.com

[12] B. Jalili, N. Aghaee, P. Jalili, and D. Domiri Ganji, "Novel usage of the curved rectangular fin on the heat transfer of a double-pipe heat exchanger with a nanofluid," *Case Stud. Therm. Eng.*, vol. 35, no. February, p. 102086, 2022, doi: 10.1016/j.csite.2022.102086.

[13] A. Gholizadeh, M. Pourfallah, M. Gholinia, M. Armin, and E. Languri, "The role of nanofluids and fins in a heat exchanger on waste energy recovery from a diesel engine: An experimental and numerical study," *Energy Reports*, vol. 8, pp. 13353–13368, 2022, doi: 10.1016/j.egyr.2022.10.009.