JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

PERFORMANCE EVALUATION OF A FLAT PLATE LOUVERED FIN HEAT EXCHANGER AND ANALYSIS OF CONDENSER

DOLLY BHASKER **SUDHIR SINGH RAJPUT** **MITHILESH SINGH**

(M.Tech. Research Scholar) (Assistant professor, RITEE) (SRU Raipur)

Abstract: Fin Heat Exchanger" investigates the efficiency of louvered fin heat exchangers in comparison to traditional plate fin heat exchangers for various heat transfer applications. The research highlights the importance of optimizing louver angles and Reynolds numbers for enhanced heat transfer. Using MATLAB for theoretical analysis, the study calculates Colburn j-factors and f-factors across different louver angles and Reynolds numbers. These theoretical values are then compared to results obtained from numerical simulations conducted using ANSYS FLUENT. The findings reveal that louvered fin heat exchangers exhibit significantly improved heat transfer rates, particularly at lower Reynolds numbers. The study identifies a louver angle of 23 degrees as optimal for maximizing heat transfer performance. However, at higher Reynolds numbers and louvered angles beyond 23 degrees, overall heat transfer efficiency diminishes due to a decrease in the Colburn j-factor. This research contributes valuable insights into the design and operation of louvered fin heat exchangers, offering guidance for more effective and efficient heat recovery systems across various industrial sectors. your research employs a combination of CAD modeling and CFD analysis to evaluate and compare the performance of air-cooled finned-tube condenser designs using different refrigerants. The use of aluminum fins and the selection of appropriate turbulence models are crucial aspects of your study. Your findings can provide insights into optimizing the design and refrigerant selection for air conditioning systems to enhance their efficiency and heat rejection capabilities.

Keywords: Colburn j- factor, Friction factor, Heat exchanger, Heat transfer, Louvered angles, CFD, Condensers, Refrigerant, ANSYS, R134.

INTRODUCTION:

Heat exchangers are crucial devices for transferring thermal energy between fluids or between fluids and solid surfaces at different temperatures, providing essential cooling and heating services in various applications. To enhance heat transfer efficiency within these devices, secondary structures known as fins are often added to their primary surfaces, considering the allowable pressure drop range. Fins come in various shapes, including rectangular, triangular, wavy, offset strip, perforated, and louvered types, and find commercial use in industries such as air conditioning, heating, power generation, automotive, and food. Recent advancements have introduced nanofluids to heat exchangers, demonstrating improved heat transfer in free, forced, and mixed convection [1-6].

The design of finned heat exchangers, showing that they outperform finless counterparts, especially at varying air velocities and flow rates. Similarly conducted experimental and numerical analyses, confirming the superiority of finned tube heat exchangers in terms of overall heat transfer efficiency. Meanwhile, analyses of heat exchangers in real-world applications, such as fuel cells, have revealed challenges in achieving expected heat transfer rates, emphasizing the need for efficient heat exchanger designs.

Louvered fins have gained prominence as efficient heat transfer components due to their compact size, light weight, and improved performance. Louvers act as obstacles, reducing the velocity of cooling air, allowing it to remain in contact with the tubes for longer periods, thereby enhancing heat transfer. Studies by researchers like Kays and London in the 1950s laid the foundation for understanding how geometric parameters affect heat transfer characteristics in louvered fins. Further research, such as the work of Beamer and Cowell, has explored varying louver angle designs, leading to improved heat rejection rates, albeit with some increase in pressure drop. The thermal-hydraulic behavior of louvered fins has been extensively investigated through numerical simulations and experimental data, solidifying their importance in enhancing heat exchanger performance.

Ansys offers a comprehensive software suite that spans the entire range of physics, providing access to virtually any field of engineering simulation that a design process requires. Organizations around the world trust Ansys to deliver the best value for their engineering simulation software investment[7-9].

METHODOLOGY:

To investigate the heat transfer performance of a louvered fin tube heat exchanger, a systematic approach is followed, which involves the development of a MATLAB code and subsequent validation against literature and simulation results. The process includes the creation of a 3D CAD model, simulations in ANSYS Fluent, and iterative trials to determine optimal louver angle and pitch for enhanced heat transfer.

MATLAB Code Development: An appropriate MATLAB code is developed to model and analyze the heat exchanger's performance based on the problem statement and governing equations.

Geometry and Dimensions: A 3D CAD model of the heat exchanger setup is created, with dimensions specified as per Table 1 in the study.

Meshing: The CAD model of the louvered fins is meshed to discretize the geometry into smaller elements. This meshed model ensures accurate simulation results and computational efficiency, as shown in Fig. 2.

Boundary Conditions: Multiple simulation trials are conducted with appropriate boundary conditions. These conditions include parameters like inlet and outlet temperatures, flow rates, and thermal properties of the fluids involved.

Optimization: The primary goal is to determine the optimum values of the louver angle and pitch that maximize heat transfer efficiency. This involves setting up the solver within the ANSYS Fluent software and computing the solution. Fig1 shows 3D model of the louvered fin heat exchanger and Meshed model of louvered fins.

Results Analysis: The results obtained from the numerical simulations are carefully examined and compared to validate their accuracy. This step may also involve comparing the outcomes with existing literature data [2-9] to ensure consistency.

Iterative Design: If the required heat transfer performance is not achieved initially, necessary modifications to the louver angle and pitch are made to the design. The analysis is then repeated iteratively until the desired results are obtained. Table 1 shows base plate dimensions for mesh generation and Table 2 shows geometric parameters of louvered fins.

This comprehensive approach, combining numerical simulations, software tools, and iterative optimization, allows for a thorough investigation of the louvered fin tube heat exchanger's performance. It ensures that the design parameters are fine-tuned to achieve the best possible heat transfer efficiency, making it a valuable contribution to the field of heat exchanger design and engineering.

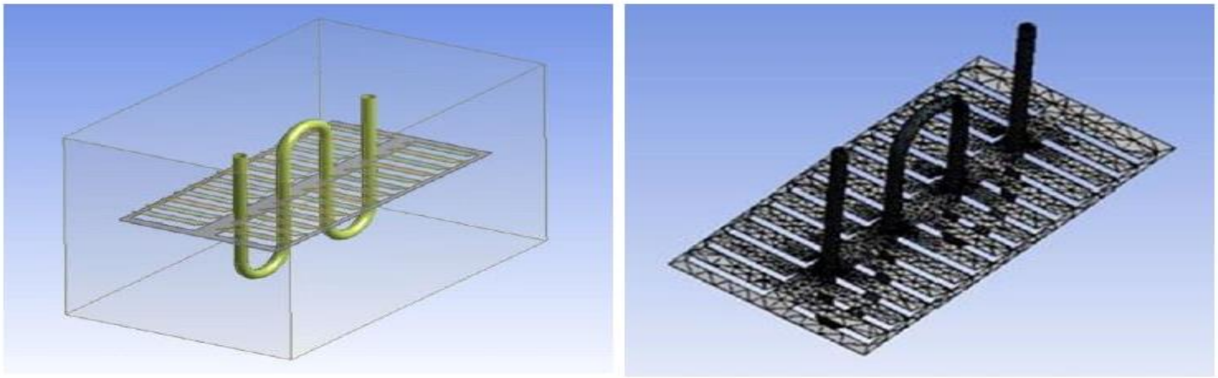
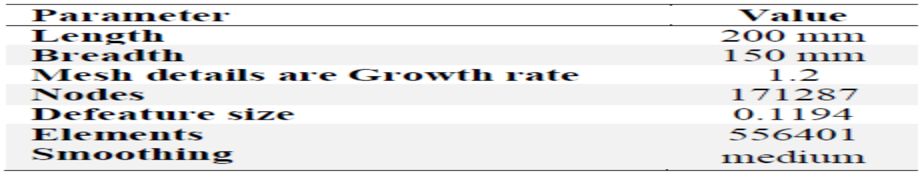


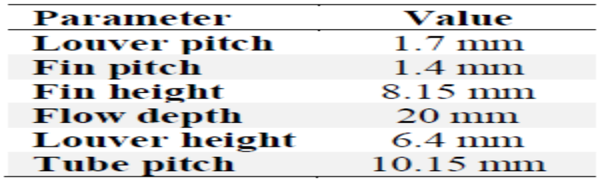
Fig1: 3D model of the louvered fin heat exchanger and Meshed model of louvered fins.

Table 1. Base plate dimension for mesh generation.



Parameter	Value
Length	200 mm
Breadth	150 mm
Mesh details are Growth rate	1.2
Nodes	171287
Defeature size	0.1194
Elements	556401
Smoothing	medium

Table 2. Geometric parameters of louvered fins.



Parameter	Value
Louver pitch	1.7 mm
Fin pitch	1.4 mm
Fin height	8.15 mm
Flow depth	20 mm
Louver height	6.4 mm
Tube pitch	10.15 mm

This analysis conducted using ANSYS FLUENT, the focus is on studying pressure drop and temperature variations in relation to the Colburn j-factor. The semi-implicit method for pressure-linked equations is employed, and the simulation adheres to an enhanced wall treatment approach, limiting the dimensionless distance y+ to values less than 5. The y+ parameter plays a crucial role in the turbulence model, serving as a non-dimensional variable based on the distance from the louver wall to the boundary layer's first node. The k-ε turbulence model is used in conjunction with enhanced wall treatment to solve for kinetic energy and dissipation as described in equations that deals with the rate of change of turbulent kinetic energy over time, accounting for its transport by advection, diffusion, production, and destruction and similarly addresses the rate of change of dissipation of turbulent kinetic energy, considering its transport by advection, diffusion, production, and destruction.

The turbulence viscosity, crucial for determining velocity, pressure, and temperature distributions through momentum and energy equations, is determined using Equation (5). The constant Cμ in this equation is set to 0.09. Fig 2.0 shows Temperature and pressure plot using R134 refrigerant and without fin condenser.

The conservation equations encompass the energy equation, continuity equation, enhanced wall-treated k-ε equations, and momentum equations. Convergence criteria are established based on normalized residual errors, which must be less than 10^-3, 10^-6, 10^-3, and 10^-3, respectively.

In terms of grid independence, it's noted that the results showed no significant variation when the number of elements exceeded 556,401 during the grid independence study. This suggests that the chosen mesh density is sufficient for obtaining accurate results in the simulation.

Discussion:

The analysis of the louvered fin heat exchanger is implemented using the pre-processing program ANSYS FLUENT, which allows for the determination of temperature and velocity distributions of the coolant and air within the heat exchanger. Figure 3 illustrates the temperature distribution of the fluid passing through the louvered fins at a Reynolds number of 100.

The temperature distribution graph reveals that the fluid temperature ranges from 318.15 K to 314.15 K within the heat exchanger. This indicates that there is an average temperature drop of 10 K compared to the initial temperature of 324 K. It's worth noting that this temperature drop is relatively low, which suggests efficient heat transfer within the louvered fin heat exchanger. In contrast, a higher temperature drop might have been expected with traditional flat plate heat exchangers.

Interestingly, the graph shows some small patches of red color at the outlet, indicating a temperature of 318 K. This unexpected result could be attributed to the possibility that the fluid at the outlet is being reheated due to the presence of hot air. This phenomenon might be a result of recirculation or mixing effects within the heat exchanger, leading to localized temperature variations. Further investigation may be needed to fully understand and optimize the heat transfer performance in this specific configuration.

In this section, the results obtained from the Computational Fluid Dynamics (CFD) analysis are discussed, focusing on temperature plots, pressure plots, and temperature differences in the context of a condenser type using R134 refrigerant, with and without fins.

Temperature Contour: The temperature contour plot for the condenser with R134 refrigerant and without fins reveals several key insights. The outer surface of the condenser reaches a temperature of 306 K. Additionally, it is observed that the highest temperature occurs near the refrigerant inlet. This temperature distribution pattern is typical in heat exchangers, where the fluid enters at a higher temperature and gradually releases heat as it flows through the tubes or fins. The temperature contour plot helps visualize how heat transfer occurs within the condenser.

Pressure Plot: The pressure plot across the condenser provides crucial information about the pressure distribution along its length. Near the refrigerant inlet, the pressure is at its maximum, with a magnitude of 155 Pa. As the refrigerant flows through the condenser coils, the pressure progressively decreases. In the next coil of tubes, the pressure drops to 103.9 Pa, and it further reduces to 11.1 Pa in the last coil of the condenser. This pressure drop is expected in a condenser, as the refrigerant releases heat and undergoes a phase change from vapor to liquid, resulting in pressure variations. Table3shows Heat flow and temperature table for both designs and refrigerants.

Overall, these results are indicative of the functioning of the condenser and its ability to transfer heat efficiently. The temperature and pressure profiles help engineers and researchers assess the performance of the condenser and make any necessary adjustments to optimize its operation, ensuring that it meets the desired cooling or refrigeration requirements.

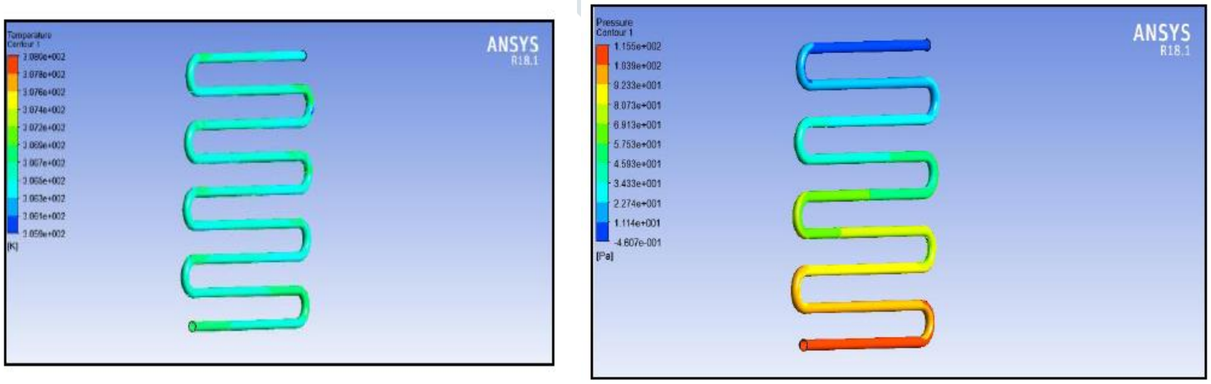
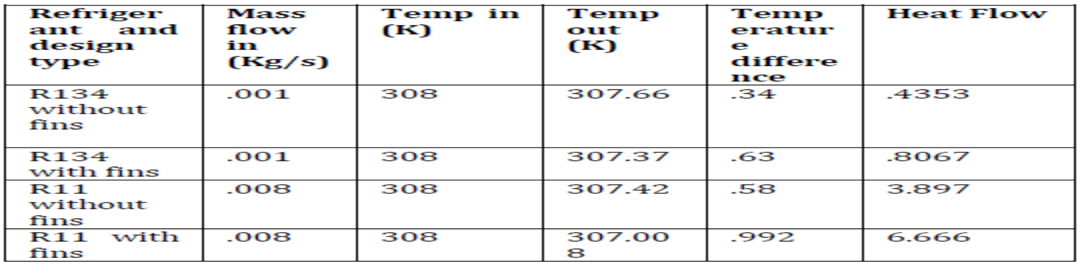


Fig2.0. Temperature and pressure plot using R134 refrigerant and without fin condenser

Table3: Heat flow and temperature table for both designs and refrigerants



Refrigerant and design type	Mass flow in (Kg/s)	Temp in (K)	Temp out (K)	Temperature difference	Heat Flow
R134 without fins	.001	308	307.66	.34	.4353
R134 with fins	.001	308	307.37	.63	.8067
R11 without fins	.008	308	307.42	.58	3.897
R11 with fins	.008	308	307.008	.992	6.666

Conclusions:

The research you've described focuses on investigating the impact of fins and refrigerant properties on the heat rejection and temperature drop characteristics of a condenser. It employs Computational Fluid Dynamics (CFD) as an alternative to conventional experimental techniques, aiming to reduce costs and time. Here's a breakdown of the key findings and implications from research:

CFD as a Viable Technique: The study suggests that CFD is a feasible and cost-effective method for analyzing condenser performance compared to traditional experimental methods. This implies that CFD can provide valuable insights without the expense and time required for physical experiments.

Effect of Rectangular Fins: The addition of rectangular fins to the condenser design results in a significant increase in the temperature drop for both R134 and R11 refrigerants. The temperature drop is approximately 90%. This suggests that the use of fins can enhance the heat transfer efficiency of the condenser.

Turbulence Model: The research employs the two-variable k-epsilon turbulence model for analysis, which is found to provide reasonable predictions of fluid flow, pressure drop, and temperature drop characteristics. This indicates the reliability of the chosen turbulence model for simulating the system.

Heat Rejection: The use of fin geometry in the condenser design, particularly for R134 refrigerant, leads to higher heat rejection compared to designs without fins. This could have practical implications for improving the efficiency of cooling systems.

Numerical Simulation Benefits: The numerical simulations conducted through CFD analysis offer valuable data to designers. This data can be used to determine optimal parameters such as Reynolds number and louver angle for louvered fin and tube type heat exchangers. This knowledge can help enhance performance without the need for expensive and time-consuming experimentation.

In summary, your research highlights the advantages of using CFD for analyzing condenser performance and demonstrates the benefits of incorporating fins into condenser designs to improve heat transfer efficiency and heat rejection. This information can be valuable for engineers and designers looking to optimize heat exchanger performance in a cost-effective manner.

References:

[1] B.A. Brigham, G.J. Vanfossen, “Length to diameter ratio and row number effects in short pin fin heat transfer”, J. Eng. Gas Turbines Power 106 (1984) 241–245.

[2] L.T. Tian, Y.L. He, Y.B. Tao, W.Q. Tao, “A comparative study on the air-side performance of wavy fin-and-tube heat exchanger with punched delta winglets in staggered and inline arrangements”, Int. J. Therm. Sci. 48 (9) (2009) 17651776.

[3] L.L. Feng, X.Z. Du, Y.P. Yang, L.J. Yang, “Study on heat transfer enhancement of discontinuous short wave finned flat tube”, Science China Technological Sciences 54 (2011) 32813288.

[4] I.H. Bell, E.A. Groll, "Air-side particulate fouling of micro channel heat exchangers: experimental comparison of air-side pressure drop and heat transfer with plate-fin heat exchanger", Appl. Therm. Eng. 31 (5) (2011) 742-749.

[5] Lijun Yang, Hui Tan, Xiaoze Du, Yongping Yang, "Thermalflow characteristics of the new wave-finned flat tube bundles in air-cooled condensers", International Journal of Thermal Sciences 53 (2012) 166-174.

[6] Pramod Kumar, Jagdish Saini, Purushottam sahu, " Numerical Analysis of Condenser Using ANSYS" (IRJET), Vol 07, issue 10, 471-478.

[7] Suhas b. Gopalakrishna, "numerical analysis on performance evaluation of a flat plate louvered fin heat exchanger, journal of engineering science and technology, vol. 17, no. 3 (2022) 1867 – 1879

[8] Ravi, R.; Senthilkumar, P.; and Mohanraj, K. (2018). Design of heat exchanger for exhaust heat recovery of a single cylinder compression ignition engine. *Journal of Engineering Science and Technology* (*JESTEC*), 13(7), 2153-2165.

[9] Ravi, R.; Pachamuthu, S.; and Kasinathan, P. (2020). Computational and experimental investigation on effective utilization of waste heat from diesel engine exhaust using a fin protracted heat exchanger. *Energy*, 200, 117489.