© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 13 Issue: 04 | July - September 2022
Paper is available at http://www.jrps.in | Email : info@jrps.in
Refereed & Double Blind Peer Reviewed

Analysis Of Thermal Performance Of A Staggered Tube Heat Exchanger With Different Fin Designs

Md Sajjad Alam[1], Shivendra Singh[2]

[1]*Research Scholar, Department of Mechanical Engineering, Corporate Institute of Science and Technology*

[2]*Assistant Professor, Department of Mechanical Engineering, Corporate Institute of Science and Technology*

ABSTRACT

Tube banks are a common structural component that may be found in heat exchangers. Smooth tube banks and finned tube banks are both quite easy to locate in the area. There are times when the flow of shell-and-tube heat exchangers, of which tube bundles are a component, is analogous to crossflow, and there are other times when the flow is analogous to longitudinal flow. A CFD model of staggered tube heat exchanger of 600 mm in length and 150 mm in breadth with a matrix of 13 tubes with a circular cross sections arranged in staggered patterns makes up the has been used for the analysis. Circular and rectangular shaped fins make up the two cases for the comparison of best performance. Longitudinal pitch (S_L) is 43.75 mm as well as transverse pitch (S_T) equals 50 mm to construct the two test portions of the cylinder with & without splitter plate. After selecting the best fin design, the longitudinal pitch (S_L) of the tubes is then varied and analysed for the best performance. The average outlet velocity showed an increase in case of finned tube heat exchangers. The maximum turbulence kinetic energy is seen in case of rectangular fins, i.e., 26.566m2/s2. The temperature contours show that the maximum average temperature at the outlet has been achieved in the case of rectangular fins, i.e., 310.969 K. The effective spacing between each pipe is 43.75 mm and the placement of rectangular fins can be an effective option for better transfer of heat in air.

Keywords: Tube banks, staggered tube heat exchanger, Circular and rectangular shaped fins, Longitudinal pitch (S_L), effective spacing

INTRODUCTION

Heat exchangers often use tube banks as a structural component. One can easily find both smooth as well as finned tube banks. The flow of shell-and-tube heat exchangers, of which tube bundles are a part, is sometimes similar to crossflow and sometimes similar to longitudinal flow. (The literature often uses tube bank to refer to a crossflow condition and bundle to refer to a longitudinal flow, although these conventions are not ubiquitous.) Boilers and condensers with tube banks have many uses in industry, and heat exchangers in furnaces are another common place for combustion to occur.

Tube banks may be arranged in one of two basic shapes, a rectangle or a rhombus, as shown in Figure 1. They are called in-line tube banks as well as staggered tube banks, respectively. The ratios of pitch to diameter, denoted a and b, are useful for distinguishing these in both the transverse as well as streamwise directions

$$a \equiv \frac{s_T}{D} \qquad b \equiv \frac{s_L}{D}$$

where “D is the cylinder diameter, sT the crosswise (transverse) pitch, and sL the streamwise pitch”. Common tube bank shapes include the in-line square (a = b), rotated square (a = 2b), and equilateral triangle (a = 2b/√3). Generally speaking, a tube bank is regarded to be compact if its product, axb <1.252, whereas a tube bank with a product of axb> 4 is considered to be widely spread.

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 13 Issue: 04 | July - September 2022
Paper is available at http://www.jrps.in | Email : info@jrps.in
Refereed & Double Blind Peer Reviewed

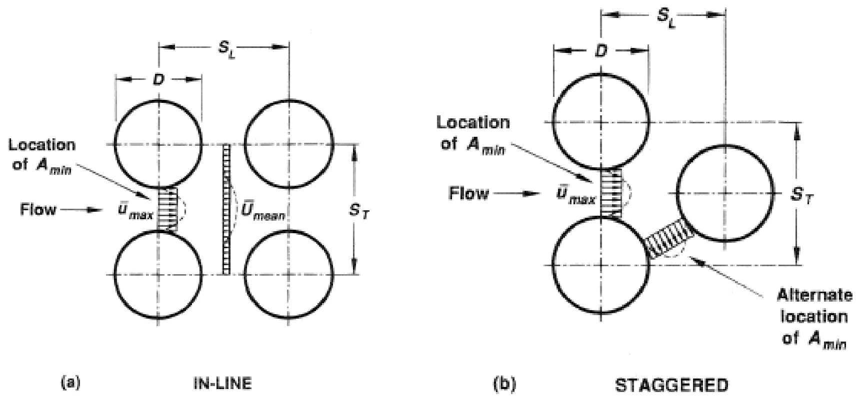


Figure 1 Schematic of (a) an in-line and (b) a staggered tube bank illustrating nomenclature, and showing location of minimum cross section

There are analytical formulas in the form of power series for the movement of an ideal fluid across in-line as well as staggered tube banks. The pressure coefficients for banks with large spacing between them have a distribution that resembles the sinusoidal pattern seen for single cylinders. For compact banks there are considerable changes in both in-line and staggered tube banks, in the latter case two extra pressure extrema may occur around φ = ±45°, where φ is the angle measured from the front of the cylinder. Flow through a single cylinder is similar to the flow of a real fluid in the body of a tube bank, however there are important distinctions. The flow The formula for the Reynolds Number, Re, is

$$Re \equiv \frac{\rho \bar{u}_{max} D}{\eta}$$

where "ρ is the fluid density, ūmax is the maximum bulk velocity i.e. the bulk velocity in the minimum cross section, D is the cylinder diameter, and η is the fluid viscosity". Stable Vortices arise behind each cylinder in a Laminar flow at a low Re, with separation happening at approximately ρ = 90°. Typically, the highest upstream flow for staggered banks occurs between the tubes immediately before it, resulting to a bifurcation of the impeding flow at the leading edge of every cylinder. There are 2 re-attachment sites for in-line banks at about φ = ±45°, and the space between cylinders is relatively dead in the wake of the cylinder before it.

For a given value of Re, vortices are released asymmetrically from each cylinder. When dealing with compact banks, the total pressure gradient causes transient motion to occur at greater Re than it would for single tubes. Streams in close proximity to one another may be in-phase, out-of-phase, or uncorrelated in their velocity, depending on the values of a and b. In staggered tube banks, the wake switches to the main flow, but in in-line banks, the shear layer between the wake and the main flow becomes unstable, allowing part of the detached vortices to be compressed by the free-stream flow.

Due to the effect of the rows that came before it, the wake becomes turbulent and the free stream turbulence becomes significant as Re continues to grow. Up to a critical Re of about 2 × 105, the Boundary Layer area is laminar, but then turbulence sets in and the separation point shifts downstream. The engineer has to be aware of two sources of vibrations in tube banks: vortex-shedding and turbulent buffeting.

As a means of improving heat transmission, roughened and otherwise improved surfaces are often used. Surface roughness is not only intentionally used, but also tends to rise naturally when smooth tubes get fouled. Increased turbulence occurs when the mean height of the roughness elements, k, is large enough. Surface roughness is said to have a greater impact in staggered banks than in in-line ones. Finned tube banks are common. Most common configurations include circular, spiral, axial, and plate-fin examples. It is usual practise to categorise tubes into "low-finned" and "high-finned" categories based on the ratio of fin height to tube diameter. Straight or tapered fins are also possible.

LITERATURE REVIEW

[1] (Khashi'ie et al., 2022) Reiner–Philippoff fluid flow across a nonlinearly contracting sheet is being investigated in this work to see how MHD as well as viscous dissipation affect radiative heat transfer. To translate the multivariable differential equation partial derivatives into the similarity equations, the proper similarity transformations need to be used. Use of the bvp4c approach is used in MATLAB software to explain the mathematical model that was generated. Tables as well as graphs are used to demonstrate the effect of various input physical characteristics on the issue. When the Eckert number as well as radiation parameter are put into the working fluid, heat transfer decreases. A higher "skin friction coefficient" and a higher local Nusselt number immediately increases the heat transfer performance by raising the magnetic parameter. It's clear that raising the suction parameter's value has an impact on the performance of the "Reiner–Philip–poff" fluid in terms of both the "skin friction coefficient" as well as the heat transfer. To verify the validity of the first solution, we must do a stability analysis which takes into account both of the possible solutions.

[2] (Nayak & Mondal, 2021) Non-Newtonian fluid flow heat transfer has not been well studied in the past because of the importance of changing viscosity. Shear stress (τ), shear rate (γ), consistency index (K), and the power-law exponent (n) are often used to explain the behaviour of the non-Newtonian fluids as $\tau=K\dot{\gamma}n$. In a non-Newtonian fluid, this is the first time that the viscosity adjustment has been postulated using boundary layer fundamentals. Mathematical predictions are backed up by experimental facts from the literature. When the viscosity adjustment is underestimated or overestimated, this results in a decrease or increase in the "effective heat transfer area". As a result of these findings, higher performance & economic advantages may be achieved via more accurate area estimations.

[3] (Kavitha et al., 2021) Copper oxide nanofluids are being tested to understand how they effect heat transmission in a "double-pipe heat exchanger" at different temperatures. A 0.004 percent volume concentration of the copper oxide nanoparticles was employed at a number of various input temperatures. As the temperature as well as the volume concentration of the nanoparticles increases, thermal performance improves, according to the research results. The thermal performance of the twin pipe heat exchanger has been the subject of a slew of investigations including nanofluids. With nanofluid, heat transmission may be improved by varying the kind of nanoparticles that are used as well as how many nanoparticles are present in a given base fluid. CuO nanoparticles in water may alter the heat transfer physiology of a double pipe heat exchanger, according to this study.

[4] (Sivalakshmi et al., 2020) Experimentally, the helical fins influence on the performance of a "twin pipe heat exchanger" has been studied. Helical fin-installed heat exchanger over inner pipe is compared to plain inner pipe heat exchangers for performance evaluation and effectiveness comparison. A hot fluid flow rate ranging from 0.01 kg/s to 0.05 kg/s as well as an intake fluid temperature of 80°C are used in this experiment. The results reveal that fins enhance the heat transfer coefficient. At increasing flow rates, the average heat transfer rate as well as heat exchanger efficacy rise to 38.46% & 35%, respectively.

[5] (Rehman et al., 2020) According to experts, non-Newtonian fluid in closed containers introduces complex mathematical models, and hence it is impossible to study the flow field in detail. It's the first numerical analysis of a non-Newtonian fluid flow fitted with a hexagonal-shaped cavity that is buoyantly convective. Hexagonal cavity has a T-shaped fin placed in the bottom wall. The adiabatic state of the hexagonal cavity's top wall is clearly visible. The bottom of the container is heated evenly. The walls on both sides of the room are maintained at a constant temperature. System of the partial differential equations is used to regulate the "buoyantly convective Casson fluid flow" around an evenly heated T-shaped fin. The numerical solution is reported using the finite element technique. Triangular as well as rectangular pieces are used to discretize the hexagonal enclosure as a computing area. The Rayleigh number is used to study the velocity as well as the temperature distribution around an evenly heated T-shape. Additionally, the dimensionless Casson fluid as well as Casson temperature are reported on a T-shaped fin's centre as well as the vertical line directions through a line graph analysis as well. Line graphs are used to show the effect of Rayleigh number on the heat transfer rate along the heated fin's surface. When the Rayleigh number is raised, we see an increase in the rate of heat transmission along the T-shaped fin's surface.

[6] (Mozafarie & Javaherdeh, 2019) An inner tube heat exchanger with the helical fins was used in this study to investigate numerically the flow as well as thermal properties of a non-Newtonian fluid. A non-Newtonian power law fluid flows through the annulus side in a laminar steady state. In order to calculate the average heat transfer coefficients

as well as pressure losses in the annulus, a 3D-CFD computational model was used. The Graetz number ($23 \times 10^3 \leq Gz \leq 55 \times 10^3$) as well as fin pitch ($25 \text{ mm} \leq p \leq 100 \text{ mm}$) are analysed numerically. Using a "smooth double pipe heat exchanger" as a test case, the model was shown to correlate well with experimentally derived correlations. Helicoidal fins created a vortex flow that improved heat transmission and pressure drop at the same time. In addition, the fin pitch was increased, which improved the thermal performance. In the industrial designs of twin pipe heat exchangers, useful and usable correlations for Nu & f are offered by data reduction.

[7] (Sharifi et al., 2018) CFD has been used to investigate the effect of the coiled wire inserts on the Nusselt number, the friction coefficient, as well as overall efficiency of twin pipe heat exchangers in this communication. The CFD softwares of the Gambit as well as Fluent were used to mesh and simulate different wire coil inserts inserted within heat exchangers. Creating structural hexahedral meshes for all heat exchanger designs was a major undertaking in order to provide trustworthy and confirmed findings. So it was made evident what objects and settings may best aid heat transfer in twin pipe heat exchangers by the verified models that were used. The results of this study show that the Nusselt values may be improved by 1.77 times by using suitable wire coils. Nusselt number as well as friction coefficient correlations for several coiled wire inserts with varying shape configurations under laminar flow were suggested as a result of the numerical simulation. Because the correlations in this study are based on the occupied areas that helical wire inserts occupy within tubes, they may be used to non-uniform helical wire insert designs, in contrast to the previous relations.

[8] (Zhang et al., 2017) The heat transmission of the non-Newtonian power law fluid through pipes with varied cross sections is examined in this work. First, a quick explanation of the non-Newtonian fluid mathematical model as well as physical model, which is separated into sections by a structured mesh. After that, numerical simulations of non-Newtonian fluid laminar heat transport in various cross sections are run. To better understand the temperature field and heat transmission in a tube, this research looks at the power-law index (n), Peclet number (Pe), and other variables based on how that field is distributed. The findings suggest that the factors we discussed have an effect on the heat transfer properties of non-Newtonian fluids.

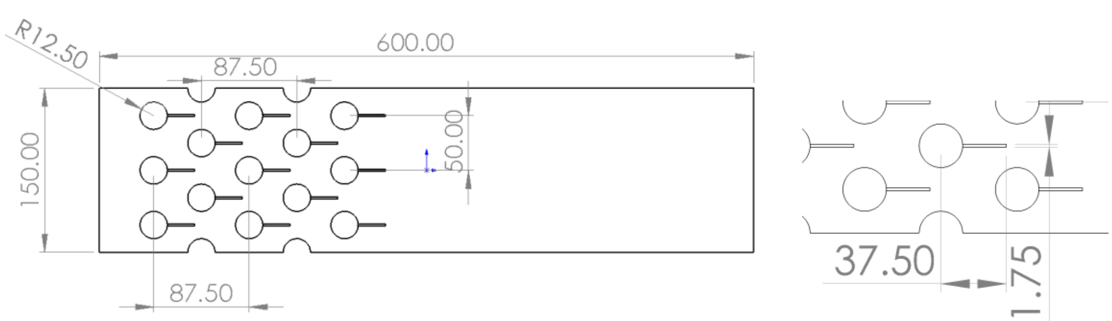
MATERIALS AND METHODOLOGY

1. Material and Design

The "staggered tube bank heat exchanger" has 13 aluminium tubes in a physical domain of 5 columns, with 4 of the tubes being fake halves. Complex issues may be reduced and treated as a two-dimensional domain to make the most use of computer resources when there is no change in the characteristics or variables in the third dimension.

Measurements of 600 mm in length and 150 mm in breadth were used in creating the SolidWorks geometry seen in Fig. 2. A matrix of 13 tubes with a circular cross sections arranged in staggered patterns makes up the computing domain, and the array consists of 5 columns of tubes. Aluminum splitter plates are fastened to the cylinders' trailing edges. Longitudinal pitch (S_L) is 43.75 mm as well as transverse pitch (S_T) equals 50 mm to construct the two test portions of the cylinder with & without splitter plate.

To achieve a (L/D) of 1, the length of the splitter plates is set to 25 mm, the thickness is set to an initial value of 1.75 mm, and the working fluid is assumed to be incompressible air.



© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 13 Issue: 04 | July - September 2022
Paper is available at http://www.jrps.in | Email : info@jrps.in
Refereed & Double Blind Peer Reviewed

Figure-2 Dimensions of the heat exchanger model

2. Methodology

A staggered tube bank heat exchanger consists of 13 aluminum tubes arranged in five columns and 4 half dummy tubes has been designed with help of SOLIDWORKS. Further converting file into .stp format. Importing the file in ANSYS design modular for performing the simulation in ANSYS. Meshing is performed on imported design. For CFD analysis fluid inlet and outlet is given at name selection. Material properties are applied to the staggered tube bank heat exchanger and the fluids. Boundary conditions are applied on the staggered tube bank heat exchanger. Thermal results are evaluated with the help of ANSYS workbench in which the modal used for this study is fluid fluent. 2 different cases of implementation of fins of different shapes, i.e. circular as well as rectangular, have been then taken. Steps 2 to 8 are then repeated again to evaluate the thermal performance of the staggered tube bank heat exchanger. Design modifications are then carried out for the best fin shape by varying the longitudinal pitch of the aluminium pitch. Steps 2 to 8 are again repeated again to evaluate the optimal thermal performance of the staggered tube bank heat exchangers.

3. Mesh Generation

The meshing tool is used to produce the necessary mesh once the two-dimensional geometry has been imported. To facilitate the meshing procedure, the area of interest is partitioned into a number of smaller subdomains. For the bare cylinder case, the produced mesh is kept at a quality of 0.87 with a skewness of 0.5, while the mesh quality for the splitter case is 0.86 with a skewness of 0.7 with total nodes 8747 and elements 8137.

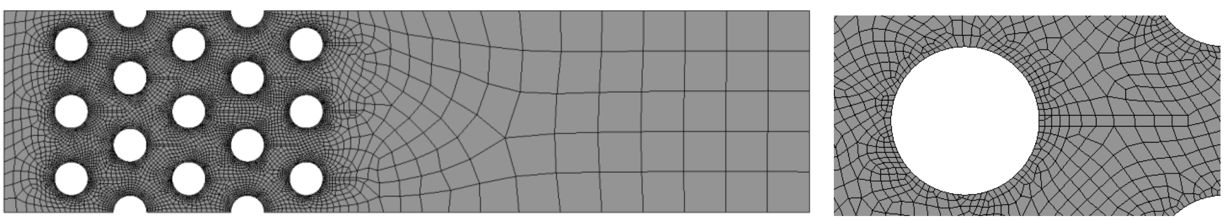


Figure-3 Mesh Generation

4. Boundary Condition

The surface temperature of the tubes is always 363 K. The air (working fluid) that enters the domain at 300 K is heated by the tube walls before being dissipated. At the region of the throat (AT), fluid velocities are at their highest.

$$V_{max} = \frac{(U \, . \, S_T)}{S_T - D}$$

(in millimetres) and D is the internal diameter of the tube (mm). For the sake of clarity, this is how we define the maximum velocity Reynolds number (Re),

$$Re = \frac{\rho \, . \, V_{max} \, . \, D}{\mu}$$

where U is the free stream velocity of air (m/s), ST is transverse pitch (mm) and D is the tube diameter (mm), ρ and μ are the air density and viscosity; respectively.

Based on the equation, the free stream velocity is computed for the Reynolds number range (5500 < Re < 14500) used for this investigation. The length (L) of the splitter plates is equal to the diameters (D) of the cylinder used,suggesting an L/D ratio range of 0.5-1.5 in order to prevent the mixing of the two vortices layers of the fluids on either side of the tube.

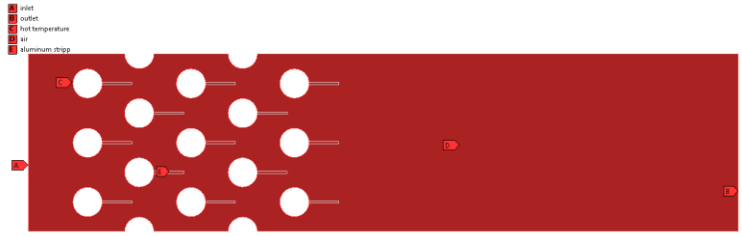


Figure 4 Boundary Conditions

- 2-D study is selected
- Energy is turned ON
- K-e model with RNG is selected
- Aluminium is selected as the strip material
- Air is selected as flowing fluid
- Velocity inlet is selected with 4.1 m/s
- Constant pressure outlet condition is selected
- Temperature of 363 K is defined at hot pipes edges

5. Implementation Of Fins

- **CASE 2: Circular Fins**

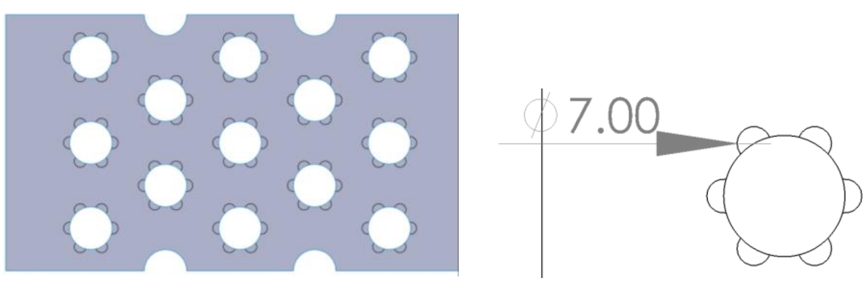


Figure -5 Case 2: Circular Fins

- **CASE 3: Rectangular Fins**

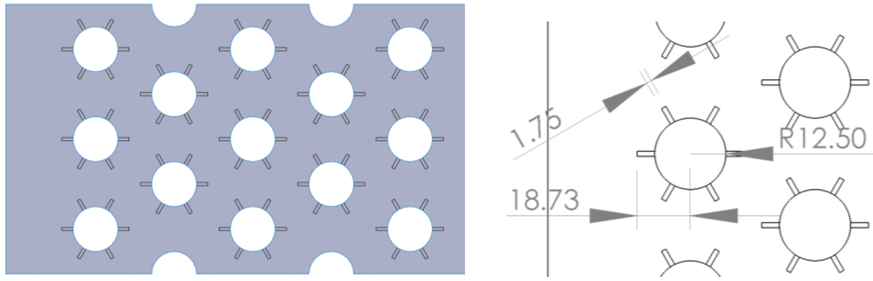
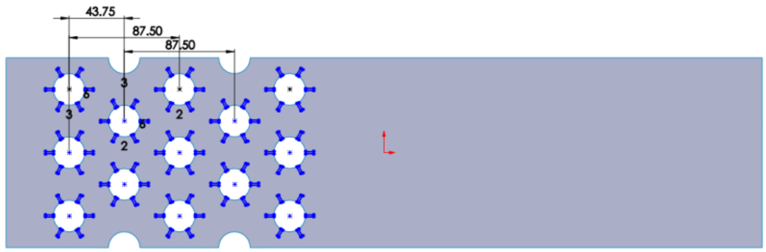


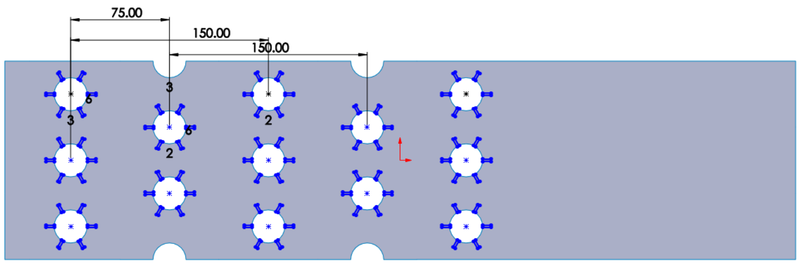
Figure -6 Case 3: Rectangular Fins

Variation In Longitudinal Space (S_L)

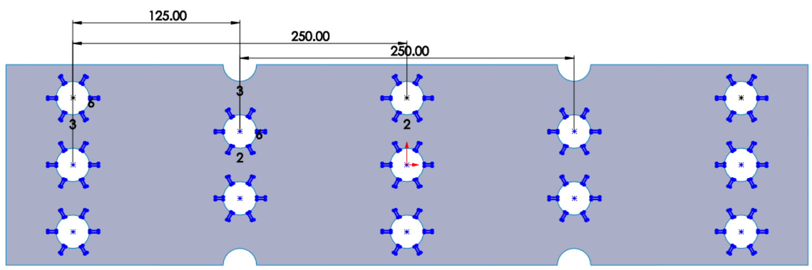
After selecting the best fin design, the longitudinal pitch (S_L) of the tubes is then varied and analysed for the best performance as shown below in the figure 7.



(a) SL=43.75



(b) SL=75

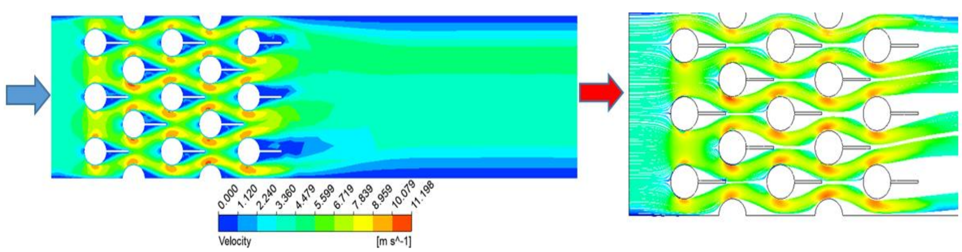


(c) SL=125

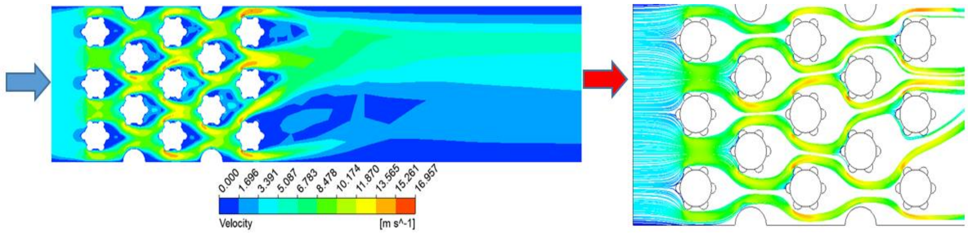
Figure 7 Temperature In Longitudinal Space (a)SL = 43.75 mm (b)SL = 75 mm (a) SL = 125 mm

RESULTS AND DISCUSSIONS

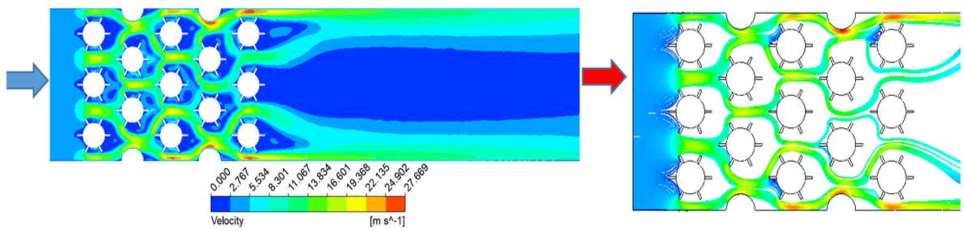
1. Velocity Variation



(a) BASE DESIGN



(b) CIRCULAR FINS

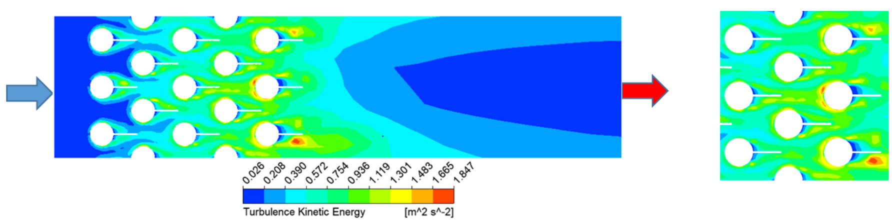


(c) RECTANGULAR FINS

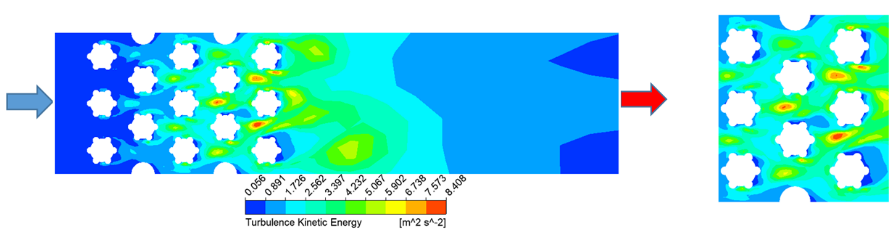
Figure -8 Velocity Contours (a) BASE DESIGN (b) CIRCULAR DESIGN (c) RECTANGULAR FINS

The velocity contours above for different fin shapes has been shown in the figure 7 along with the maximum velocities for the different fin shapes along the length of the staggered tube heat exchanger. Given the same inlet velocity in all the cases, the mainstream velocity curves first show an abrupt increase in all the cases in small regions while passing between the tubes, then it decreases at the outlet, as can be clearly observed. However, the boundary layer effect is increased in the case of circular and rectangular fins and the overall velocity of the mainstream received at the outlet is lower than that of the initial design.

2. Turbulence Kinetic Energy Variation

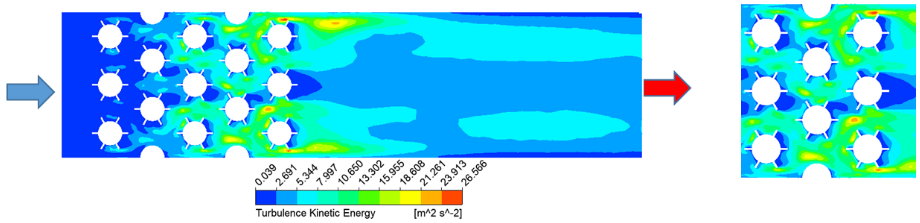


(a) BASE DESIGN



(b) CIRCULAR FINS

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 13 Issue: 04 | July - September 2022
Paper is available at http://www.jrps.in | Email : info@jrps.in
Refereed & Double Blind Peer Reviewed

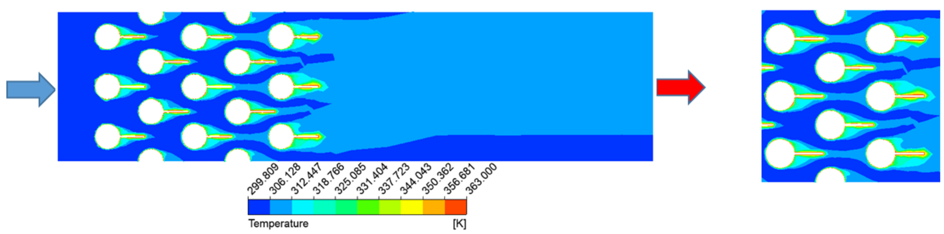


(c) RECTANGULAR FINS

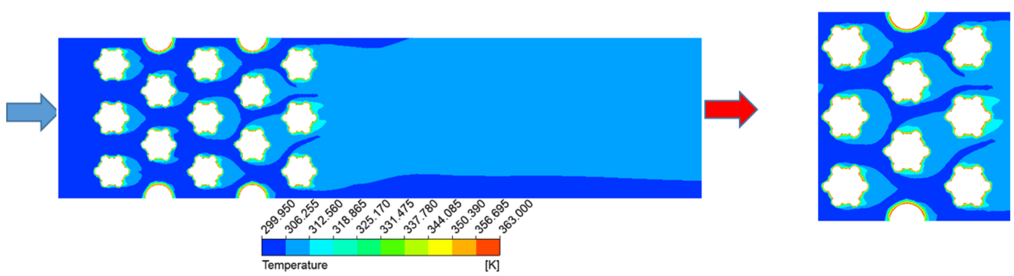
Figure -9 Velocity Contours (a) BASE DESIGN (b) CIRCULAR DESIGN (c) RECTANGULAR FINS

The turbulence kinetic energy values also show an increase with the shape of fins, due to the fact that the fin shapes provide greater obstacle to the flow and thus causing more turbulence. The maximum turbulence kinetic energy is seen in case of rectangular fins, i.e., 26.566m2/s2. But this value only corresponds to the regions between the tubes, after which it decreases. However, the average turbulence kinetic energy for circular as well as rectangular fins are greater than the base design.

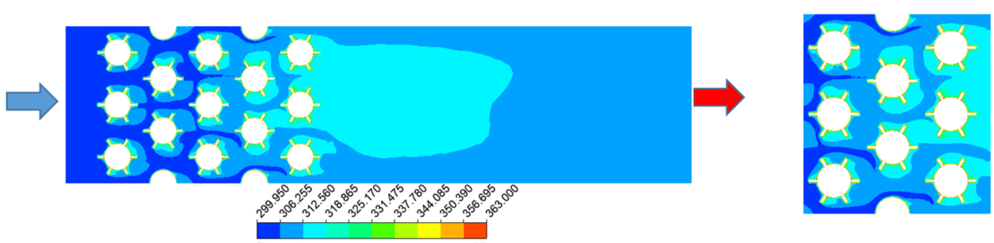
3. Temperature Variation



(a) BASE DESIGN



(b) CIRCULAR FINS



(c) RECTANGULAR FINS

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 13 Issue: 04 | July - September 2022
Paper is available at http://www.jrps.in | Email : info@jrps.in
Refereed & Double Blind Peer Reviewed

Figure -10 Velocity Contours (a) BASE DESIGN (b) CIRCULAR DESIGN (c) RECTANGULAR FINS

Table 1 Average temperature at the outlet

CASE	AVG. OUTLET TEMPERATURE
Base design	305.901 [K]
Circular fins	306.732 [K]
Rectangular fins	310.969 [K]

The temperature contours show that the maximum average temperature at the outlet has been achieved in the case of rectangular fins, i.e., 310.969 K. The temperatures of all the 3 cases along the line shown in figure 10, have been compared graphically in the figure 11.

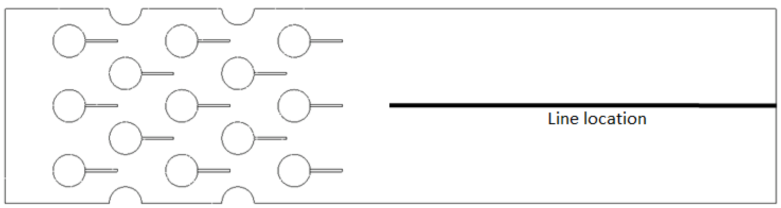


Figure-11 The Reference Line

Figure 11 compares the temperatures for the different fin shapes along the length of the staggered tube heat exchanger. While the temperature curve in the case of the base design and the circular fins almost remain constant, the rectangular fin case shows a declining curve along the reference line. However, the overall temperature along the reference line at each point and also at the outlet is maximum in the case of the rectangular fins.

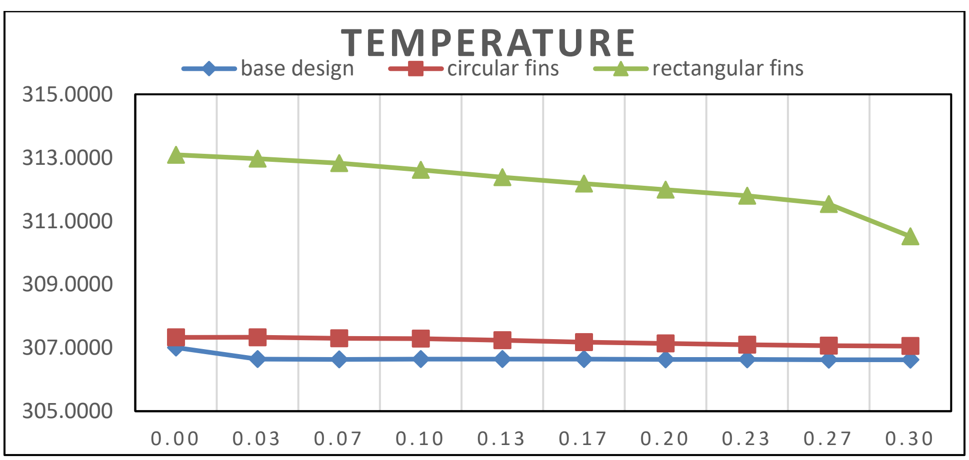
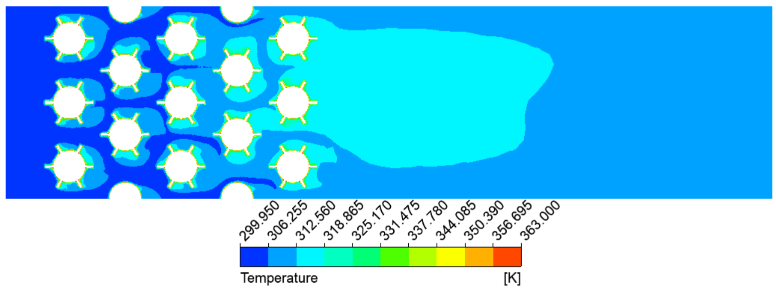


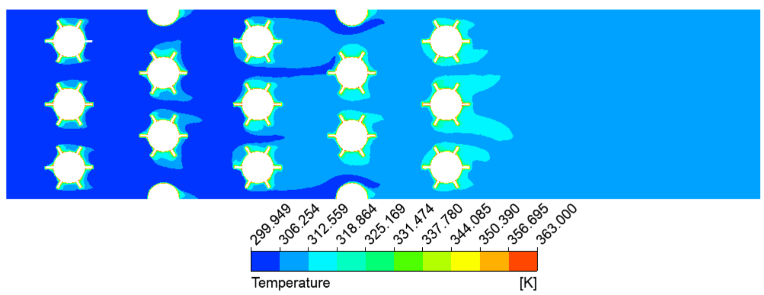
Figure 12 Temperatures along the reference line for all the 3 cases

4. Variation Of Temperature In Longitudinal Space (S_L)

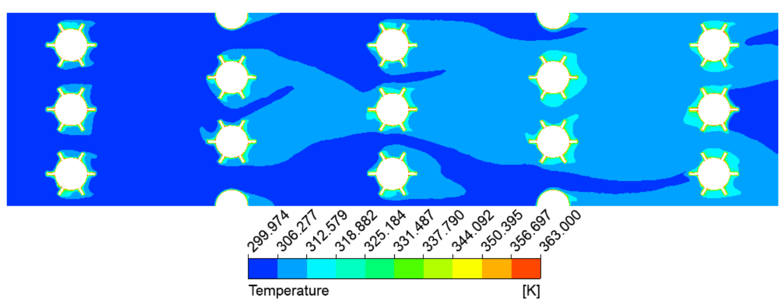
© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 13 Issue: 04 | July - September 2022
Paper is available at http://www.jrps.in | Email : info@jrps.in
Refereed & Double Blind Peer Reviewed



(a) S_L = 43.75 mm



(b) S_L = 75 mm



(c) S_L = 125 mm

Figure 13 Temperature In Longitudinal Space (a)SL = 43.75 mm (b)SL = 75 mm (a) SL = 125 mm

Table 2 Average temperature at the outlet

longitudinal spacing (SL)	AVG. OUTLET TEMPERATURE
43.75 mm	310.969 [K]
75 mm	308.640 [K]
125 mm	306.381 [K]

EFFECTIVENESS OF HEAT EXCHANGER

Effectiveness is calculated with respect to cold inlet and outlet temperature.

Table 3 Effectiveness for different fin shapes

CASE	Inlet cold temperature	Maximum hot temperature	OUTLET TEMPERATURE	Effectiveness
Base design	300.15 [K]	363.15 [K]	305.901 [K]	0.091
Circular fins	300.15 [K]	363.15 [K]	306.732 [K]	0.104
Rectangular fins	300.15 [K]	363.15 [K]	310.969 [K]	0.172

Table 4 Effectiveness for different longitudinal pitches

CASE	Inlet cold temperature	Maximum hot temperature	OUTLET TEMPERATURE	Effectiveness
$S_L = 43.75$	300.15 [K]	363.15 [K]	310.969 [K]	0.172
$S_L = 75$	300.15 [K]	363.15 [K]	308.640 [K]	0.135
$S_L = 125$	300.15 [K]	363.15 [K]	306.381 [K]	0.099

The effective spacing between each pipe is 43.75 mm and the placement of rectangular fins can be an effective option for better transfer of heat in air.

CONCLUSION

A staggered tube heat exchanger of 600 mm in length and 150 mm in breadth has been used for the analysis. A matrix of 13 tubes with a circular cross sections arranged in staggered patterns makes up the computing domain, and the array consists of 5 columns of tubes. Aluminum splitter plates are fastened to the cylinders' trailing edges. After selecting the best fin design, the longitudinal pitch (S_L) of the tubes is then varied and analysed for the best performance. Given the same inlet velocity in all the cases, the mainstream velocity curves first show an abrupt increase in all the cases in small regions while passing between the tubes, then it decreases at the outlet, as can be clearly observed. However, the boundary layer effect is increased in the case of circular and rectangular fins and the overall velocity of the mainstream received at the outlet is lower than that of the initial design. The maximum turbulence kinetic energy is seen in case of rectangular fins, i.e., 26.566m2/s2. But this value only corresponds to the regions between the tubes, after which it decreases. However, the average turbulence kinetic energy for circular as well as rectangular fins are greater than the base design. The temperature contours show that the maximum average temperature at the outlet has been achieved in the case of rectangular fins, i.e., 310.969 K. The effective spacing between each pipe is 43.75 mm and the placement of rectangular fins can be an effective option for better transfer of heat in air.

REFERENCES

[1] N. S. Khashi'ie, I. Waini, A. R. M. Kasim, N. A. Zainal, A. Ishak, and I. Pop, "Magnetohydrodynamic and viscous dissipation effects on radiative heat transfer of non-Newtonian fluid flow past a nonlinearly shrinking sheet: Reiner–Philippoff model," *Alexandria Eng. J.*, vol. 61, no. 10, pp. 7605–7617, 2022, doi: 10.1016/j.aej.2022.01.014.

[2] S. K. Nayak and S. Mondal, "Viscosity correction in convective heat transfer correlation of non-Newtonian fluid pipe flow: Revisited," *Chem. Eng. Sci.*, vol. 235, p. 116472, 2021, doi: 10.1016/j.ces.2021.116472.

[3] R. Kavitha, Y. Methkal Abd Algani, K. Kulkarni, and M. K. Gupta, "Heat transfer enhancement in a double pipe heat exchanger with copper oxide nanofluid: An experimental study," *Mater. Today Proc.*, 2021, doi: https://doi.org/10.1016/j.matpr.2021.11.096.

[4] S. Sivalakshmi, M. Raja, and G. Gowtham, "Effect of helical fins on the performance of a double pipe heat exchanger," *Mater. Today Proc.*, vol. 43, no. xxxx, pp. 1128–1131, 2020, doi: 10.1016/j.matpr.2020.08.563.

[5] K. U. Rehman, M. Y. Malik, Q. M. Al-Mdallal, and W. Al-Kouz, "Heat transfer analysis on buoyantly convective non-Newtonian stream in a hexagonal enclosure rooted with T-Shaped flipper: Hybrid meshed analysis," *Case Stud. Therm. Eng.*, vol. 21, no. April, p. 100725, 2020, doi: 10.1016/j.csite.2020.100725.

[6] S. S. Mozafarie and K. Javaherdeh, "Numerical design and heat transfer analysis of a non-Newtonian fluid flow for annulus with helical fins," *Eng. Sci. Technol. an Int. J.*, vol. 22, no. 4, pp. 1107–1115, 2019, doi: 10.1016/j.jestch.2019.03.001.

[7] K. Sharifi, M. Sabeti, M. Rafiei, A. H. Mohammadi, and L. Shirazi, "Computational fluid dynamics (CFD) technique to study the effects of helical wire inserts on heat transfer and pressure drop in a double pipe heat exchanger," *Appl. Therm. Eng.*, vol. 128, pp. 898–910, 2018, doi: 10.1016/j.applthermaleng.2017.08.146.

[8] H. Zhang, Y. Kang, and T. Xu, "Study on Heat Transfer of Non-Newtonian Power Law Fluid in Pipes with Different Cross Sections," *Procedia Eng.*, vol. 205, pp. 3381–3388, 2017, doi: 10.1016/j.proeng.2017.09.845.