 ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

CFD Analysis of Heat Transfer Enhancement in a Heat Exchanger Using Various Baffle Arrangements

[1]Ankit Uppal, [2]Dr. Vinod Kumar, [3]Dr. Chanpreet Singh
[1]Studied at Punjabi University, Patiala, Punjab, India
[2,3]Dept. of Mechanical Engineering, Punjabi University, Patiala, Punjab, India

Abstract
The present study reports the heat transfer enhancement in a heat exchanger tube by installing seven different baffle arrangements. The purpose of the study is to find out the optimum baffle shape and arrangement. For this analysis, a three dimensional finite volume based CFD tool ANSYS 14.5 Fluent was used. A heat exchanger tube of length 0.1m and tube diameter 0.01m which was made of Al and Cu has been considered. All the calculations were carried out at 10,000 Reynolds number. Boundary conditions were defining with appropriate material property in fluent software. In the solver, all flows were specified as steady state and incompressible. The realizable k-ε turbulence model with standard wall function was set for all models for turbulent flow. According to results, it concluded that in case of single baffle used, rate of heat transfer is maximum for rectangular shape baffle surface and in case of baffle combinations; rate of heat transfer is maximum for rectangular and triangular baffle. The reason behind maximum heat transfer rate was that due to use of baffles, turbulence was increased as they allow more mixing of fluid layers and resulted in increase of heat transfer through the heat exchanger tube.

Keywords
Heat Exchanger Tube, Turbulent Flow, CFD, Fluent, Simulation, Turbulence

I. Introduction

Heat exchangers are used in a wide range of engineering applications, such as power generation, auto and aerospace industry, electronics and HVAC. The main purpose of a heat exchanger is to efficiently transfer the heat from one fluid to the other, which in most of the cases is separated by a solid wall. In many applications such as, power generation and HVAC, the efficiency of the heat exchangers play an important role in controlling the overall performance of the system. An efficient heat exchanger in such systems could result in the lesser consumption of the energy resource, which provides both economical and environmental benefits. The performance of heat exchanger can be improved by enhancing the heat transfer between the heat exchanger fluids. The objective of heat transfer enhancement can be achieved by increasing the surface heat transfer coefficient through improving the thermal contact of the heat exchanger fluid with the wall. There are numerous ways to increase the heat transfer which include treated surfaces, rough surfaces, extended surfaces, surface vibration, fluid vibration and jet impingement. In recent years, there has been great demand for high performance, lightweight, compact and economical heat transfer components. The fins are recognized as one of the most effective means of increasing the heat dissipated. Baffles are commonly used to facilitate the dissipation of heat from a heated wall to the surrounding environment. Baffles are used to enhance convective heat transfer in a wide range of engineering applications, and offer a practical means for achieving a large total heat transfer surface area without the use of an excessive amount of primary surface area. Baffles are commonly applied for heat management in electrical appliances such as computer power supplies or substation transformers. Other applications include IC engine cooling, such as baffles in a car radiator. The objective of this study was to find out optimum type of baffle and arrangement used for maximum heat transfer rate. This task was performed by using CFD as a tool. Seven basic shapes of baffles will be used in this study to find out best shape. Maximizing the heat transfer will be main criteria for selection of optimum fin.

II. Geometrical Modeling and Mesh Generation

A. Methodology

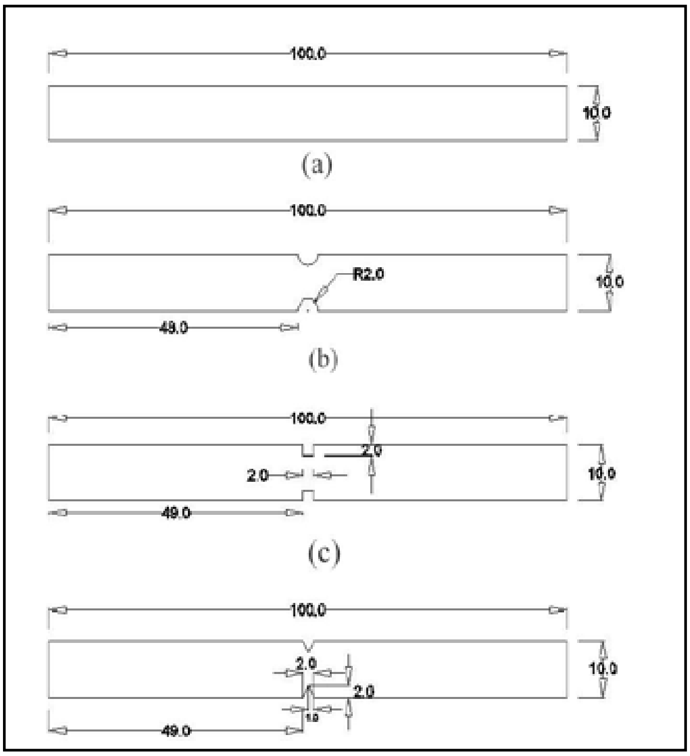
For CFD simulation, first of all, geometry of the various baffle arrangements was created using ANSYS 14.5 Fluent (fluid flow). The geometry of the heat exchanger tube is in 2D view. After geometry creation next step is to mesh the geometrical model, which was also done in the same software. After that naming the different sections of the tube like inlet, outlet, wall and center line in the ANSYS software was done. Finally meshing is created. It contains Tetrahedral and Hexahedral cells having both triangular and quadrilateral faces at the boundaries. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed. Then dimensional units for CFD domain are specified. In FLUENT desired turbulence model was selected for modeling on the basis of literature review. After selection of turbulence model boundary conditions are specified. Fluent has capability to store value of physical parameters for any point in the domain for analysis. Seven points were created to store the value of physical parameters such as temperature, velocity and pressure. FLUENT is now ready to simulate flow problem. Finally, post processing was done for result analysis.

B. Geometric Modeling

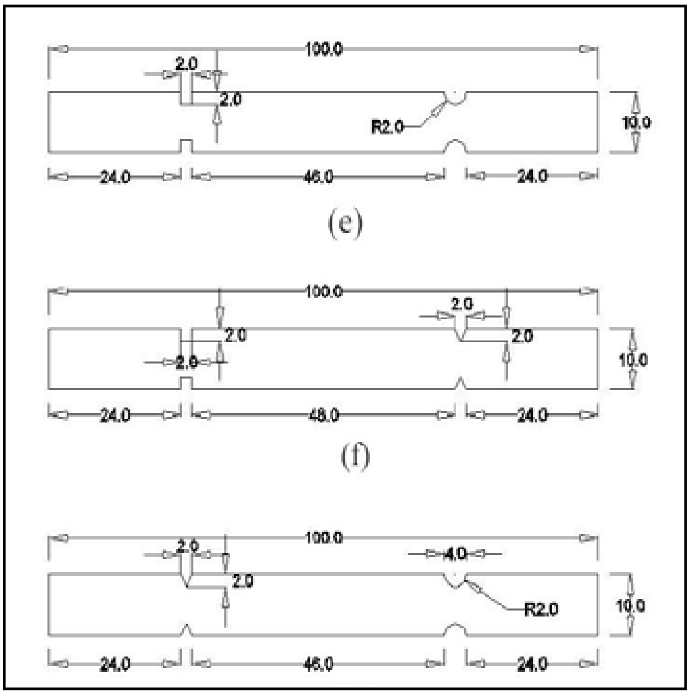
Geometry is simplified by considering the plane symmetry. Geometry of the heat exchanger tube is created in 2D. The symmetry of the geometry allows to simulate half of the tube diameter using axisymmetric boundary condition, which reduced the computational efforts and the simulation time. For all the cases, the impact of the baffle height on the heat transfer enhancement and flow characteristics was investigated for baffle height, which was 2 cm. This corresponds to the flow area reduction of 40% at the baffle. Seven different baffles were considered in this study. In figure 1 shows 2-D geometry of various baffle arrangements. All the dimensions are in 'cm'. The thickness of the tube is 1cm/1 m.

C. Meshing of CFD Domain

After making geometry of the CFD domain, next step is to mesh the domain. To perform better results using CFD tool it was mandatory to use better quality of mesh. Initially a relatively coarser mesh is generated with 1.8 Million cells. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries as shown in fig. 2. Care is taken to use structured cells (Hexahedral) as much as possible, for this reason the geometry is divided into several parts for using



(a)



(b)

Fig. 1: Geometry of Different Types of Baffle Arrangements

automatic methods available in the ANSYS meshing client. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed.

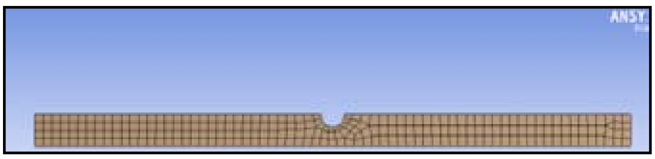


Fig. 2: Meshing of the Tube

After meshing naming of different sections of the tube like inlet, outlet, wall and center line in the ANSYS software is done one by one as shown in fig. 3.

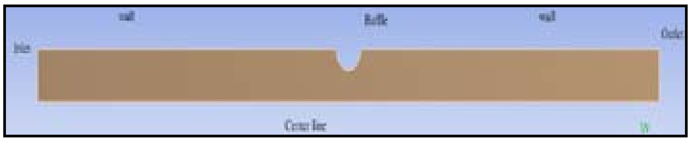


Fig. 3: Named Selections of Various Faces

III. Boundary Conditions

After mesh generation, boundary conditions are defined for CFD domain as shown in Table 1. Specify "boundary condition" icon is used to create boundaries. In FLUENT launcher, both fluid and solid can be defined. Bottom most wall is defined as the center line around which the tube is axi-symmetric. The walls are separately specified with respective boundary conditions. 'No slip' condition is considered for each wall. Each wall is set to zero heat flux condition. In the boundary conditions, the value of Turbulent Kinetic Energy and Turbulent Dissipation Rate is taken as constant. Generally, the materials used in this analysis are Aluminum and Copper. The fluid used in this analysis is air. The material and fluid properties of both the materials are shown in Table 2.

The viscosity of air is 0.0120568824 m^2/s.

Table 1: Boundary Conditions of the Model

	Zone	Type	T (K)	Other Specifications
Boundary Conditions	Center Line	Axis	-	-
	Interior Surface	Interior	-	-
	Wall	Wall	375	Stationary Wall Wall Roughness Constant = 0.5
	Inlet	Velocity-Inlet	300	Velocity magnitude = 1.46 m/s
	Outlet	Pressure-Outlet	300	-

Table 2: Material and Fluid Properties

Material / Fluid	Density (Kg/m^3)	Specific Heat (J/kgK)	Thermal Conductivity (W/mk)
Aluminum	2719	871	202.4
Copper	8920	385	401
Air	1.225	1006.43	0.0242

IV. CFD Simulation

In this work, FLUENT software is used for simulation. Main focus of this work is on heat transfer analysis of circular tube having various baffle arrangements. In CFD simulation, selection of turbulence model is an important issue. Although we go for selecting the model to be used in which we turn the energy equation on as well as we

select the turbulence model out of all the turbulence models because according to the literature review this turbulence model most accurately predicts the flow behavior due to the use of baffles.

 ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

A. Governing Equations and Turbulence Modeling

The governing equations for fluid dynamics are conservation equations for mass, momentum and energy. The governing equations have actually been known for over 150 years. In the 19th century two scientists, Navier and Stokes described the equations for a viscous, compressible fluid, which are now known as the Navier-Strokes Equations. These equations form a set of differential equations.

$\partial/(\partial t)\ (\rho\phi) + div(\rho\nabla\phi - \Gamma_\phi\ grad\phi) = S$

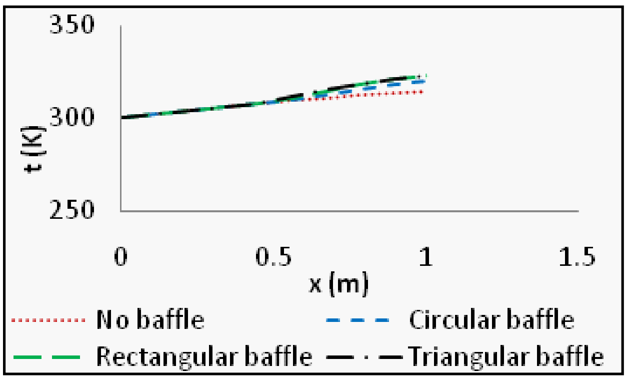
V. Results and Discussion

By completion of all the test runs in Fluent, several key performance indicators were studied to understand the heat transfer characteristics and trends for each baffle configuration. To understand results we study temperature based results in graphical mode and velocity results.

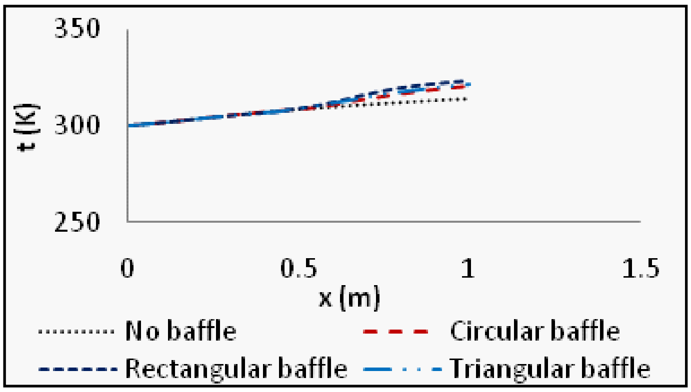
A. Temperature Variation Along the Center Line

Fig. 4 provides a temperature variation along the center line of the tube. Here the baffle is placed at 0.048 m. At inlet, the temperature in all the cases is 300 K. As we proceed along the baffle, the temperature of air increases slightly. At x=0.05m, the temperature of air in case of no baffle is 309.21 K and 309.09 K, 309.20 K and 309.33 K in case of circular baffle, 310.65 K and 310.36 K in case of rectangular baffle, and 311.11 K and 310.48 K in case of triangular baffle. Beyond the baffle, the temperature of air goes on increasing. At x=0.1 m, the temperature in case of no baffle is 314.10 K and 313.87 K, 320.33 K and 320.21 K in case of circular baffle, 323.06 K and 323.14 K in case of rectangular baffle, and 322.62 K and 321.66 K in case of triangular baffle.

Fig. 5 provides a temperature variation along the center line of the tube. Here the baffle is placed at 0.025 m and 0.074 m. At inlet, the temperature in all the cases is 300 K. As we proceed along the baffle, the temperature of air increases slightly. At x=0.03 m, the temperature of air in case of no baffle is 305.99 K and 305.87 K, 308.25 K and 308.01 K in case of rectangular & circular baffle, 307.17 K and 307.18 K in case of rectangular & triangular baffle, and 307.69 K and 307.69 K in case of triangular & circular baffle. Beyond the baffle, the temperature of air goes on increasing. At x=0.7 m, the temperature in case of no baffle is 311.85 K and 311.70 K, 319.02 K and 319.08 K in case of rectangular & circular baffle, 320.41 K and 320.46 K in case of rectangular & triangular baffle, and 319.81 K and 319.63 K in case of triangular & circular baffle. Beyond that the temperature of air increases. At x=0.1 m, the temperature in case of no baffle is 314.12 K and 313.87 K, 324.74 K and 324.88 K in case of rectangular & circular baffle, 326.38 K and 326.40 K in case of rectangular & triangular baffle, and 325.71 K and 325.65 K in case of triangular & circular baffle.

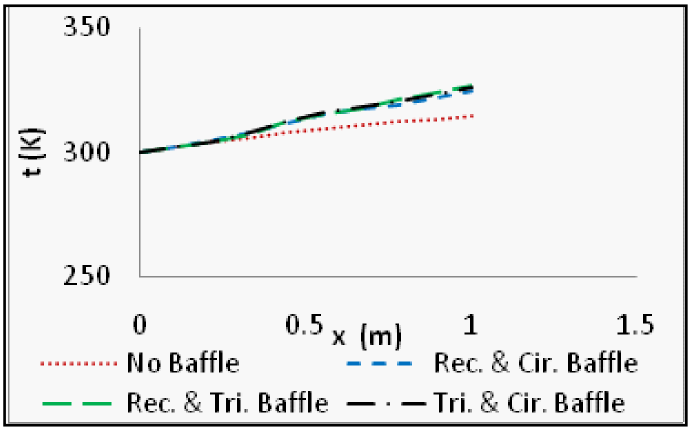


(a)

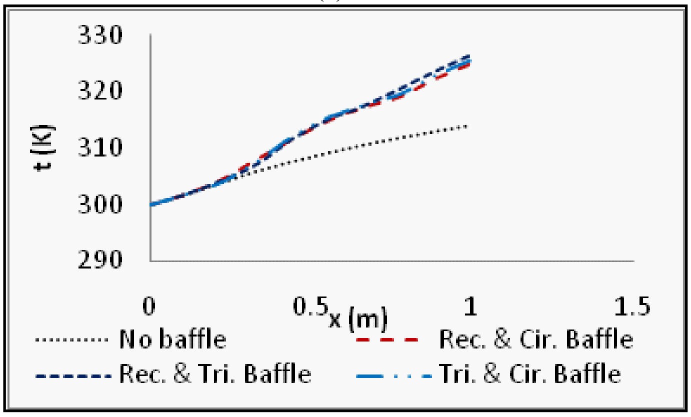


(b)

Fig. 4: Temperature Variations Along the Center Line in Case of Single Baffle Used W.R.T No Baffle (a) Al, (b) Cu



(a)



(b)

Fig. 5: Temperature Variations Along the Center Line in Case of Double Baffle Used W.R.T no Baffle (a) Al, (b) Cu

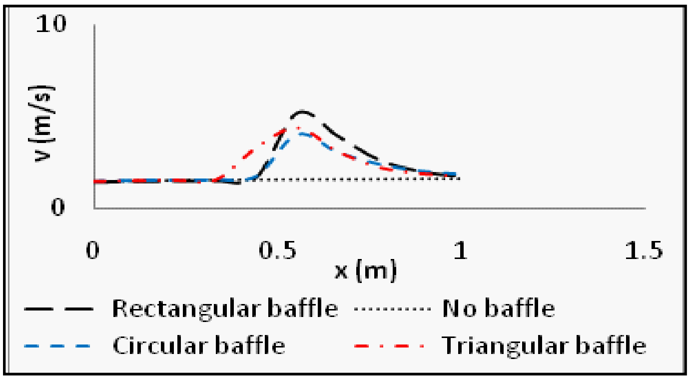
There is drop in temperature whenever there is a baffle in place along the centerline. This is because of more turbulence created by the baffle inserts in the path of fluid flow and fluid layers mingle with each other thereby causing more heat transfer, also there is increase in surface area.

B. Velocity Variation Along the Center Line

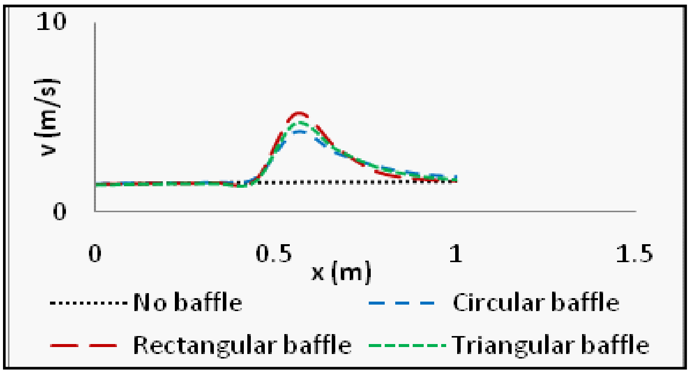
Velocity variation plots give an idea of flow separation at the places where baffles are being placed along the centerline of the heat exchanger tube as shown in fig. 6 and 7. This is because at the places along the length of the tube where baffles are being placed, there is more turbulence created by the vortex generators (baffles) which results in more mixing of the fluid layers.

Due to this, the temperature drops where a baffle is placed which results in decrease of viscosity and hence decrease in pressure.

Since the flowing fluid here follows Bernoulli's equation. So decrease in pressure results in sharp increase in velocity and hence flow separation occurs.

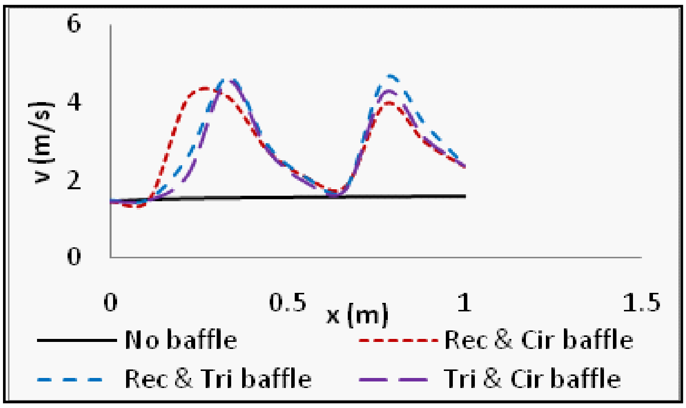


(a)

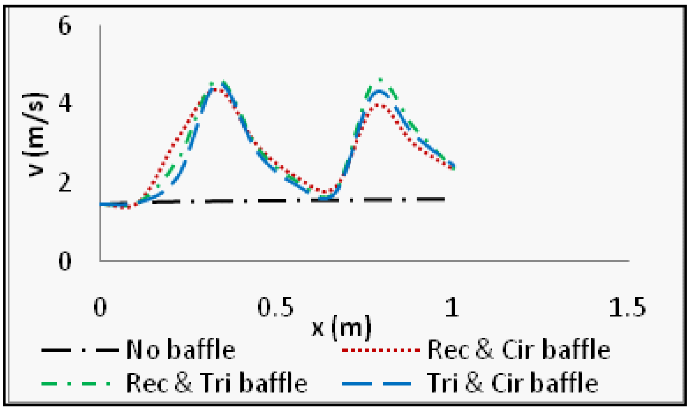


(b)

Fig. 6: Velocity Variations Along the Center Line in Case of Single Baffle Used W.R.T no Baffle (a) Al, (b) Cu



(a)

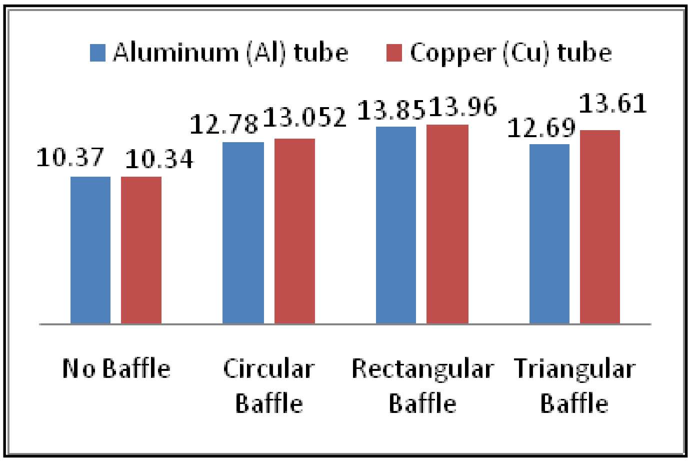


(b)

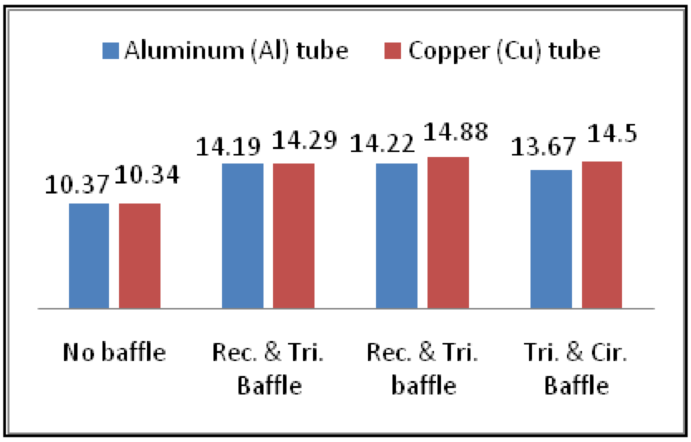
Fig. 7: Velocity Variations Along the Center Line in Case of Single Baffle Used W.R.T no Baffle (a) Al, (b) Cu

C. Comparison of Different Baffle Arrangements on the Basis of in Both Cases

Due to increased turbulence, there is increase in value of heat transfer coefficient (h) along the length of the tube wherever a baffle is placed. The values of average heat transfer coefficient for different baffle and their combinations is listed in the table 3 below along with the percentage increase in the heat transfer coefficient "h". Without baffle, the heat transfer coefficient is found to be 10.37 W/m2K in case of Al tube and 10.34 W/m2K in case of Cu as shown in figure 8. Maximum increase in the value of average heat transfer coefficient is observed in the case when rectangular and triangular baffle is used with an increase of 27.07 % (in case of Al Tube) and 31.77 % (in case of Cu Tube) in the value of average heat transfer coefficient. Whereas in case of single baffle used, the maximum increase in is observed in case when rectangular baffle is used with an increase of 27.23 % (in case of Al tube) and 25.93 % (in case of Cu tube).



(a)



(b)

Fig. 8: Comparison of Al and Cu Tube With Different Baffle Arrangement

 ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

Table 3 Comparison of % increase in avg 'h' between Al and Cu tube

S.No.	Type of baffle Used	Average value of "h" (Al)	Percentage increase in "h" (Al)	Average Value of "h" (Cu)	Percentage increase in "h" (Cu)
1	No Baffle	10.37		10.34	
2	Circular Baffle	12.78	19.31 %	13.05	20.77 %
3	Rectangular Baffle	13.85	27.23 %	13.96	25.93 %
4	Triangular Baffle	12.69	18.28 %	13.61	24.03 %
5	Rectangular & circular Baffle	14.19	26.92 %	14.29	27.64 %
6	Rectangular & triangular Baffle	14.22	27.07 %	14.88	31.77 %
7	Triangular & Circular Baffle	13.67	24.14 %	14.50	28.69 %

VI. Conclusion

The following conclusions have been drawn from the analysis:

1. Without baffle, the heat transfer coefficient was found to be 10.37 W/m2K in case of Al tube and 10.34 W/m2K in case of Cu.
2. In case of single baffle used, maximum increase in the value of average heat transfer coefficient is observed in the case when rectangular baffle is used with an increase of 27.23 % (in case of Al tube) and 25.93 % (in case of Cu tube) w.r.t no baffle.

Value of heat transfer coefficient "h" increases with different baffle combinations rather than using a single baffle.

3. Maximum increase in the value of average heat transfer coefficient is observed in the case when rectangular and triangular baffle is used with an increase of 27.07 % (in case of Al Tube) and 31.77 % (in case of Cu Tube) w.r.t no baffle.

Thus increase in the value of heat transfer coefficient means increase in heat transfer rate through the tube of heat exchanger and better performance of the heat exchanger.

References

[1] S.R Mcilwain,"Improved prediction methods for finned tube bundle heat exchangers in cross flow", Ph.D Thesis, University of Strathclyde, Glasgow, 2003.

[2] Incropera F. P., DeWitt D. P., "Fundamentals of heat and mass transfer", 4th Edition, John Wiley & Sons, 1996, pp. 147-172.

[3] P. K. Nag,"Heat and Mass Transfer", 2nd Edition, Tata McGraw Hill Co., 2006, pp. 86-108 & 425-449.

[4] J. P. Holman, "Heat Transfer", 9th Edition, Tata McGraw Hill Co, 2004, pp: 43-53 & 315-350.

[5] Yunus A. Cengel,"Heat transfer- A Practical Approach" SI units 2nd Edition, Tata McGraw Hill Co., 2004, pp. 156-168, 333-352 & 459-500.

[6] X.Liu, M.K. Jensen,"Geometry Effects on Turbulent Flow and Heat Transfer in Internally Finned Tubes", ASME J. of Heat transfer, Vol. 123, 2001, pp: 1035-1044.

[7] Petukhov, B., Kurgano, V., Gladuntsov, A., "Heat transfer in turbulent pipe flow of gases with variable properties", Heat Transfer Sov. Res. 5, 1973, pp: 109-116.

[8] Lin. C.N., Jang, J.Y., "Conjugate heat transfer and fluid flow analysis in fin tube heat exchangers with wave-type vortex generators", J. Enhanced Heat Transfer, 2002, pp: 123-136.

[9] Gnielinski, V.,"New equations for heat transfer in turbulent pipe and channel flow", Int. Chem. Eng., 1976, pp: 359-368.

[10] Fiebig, M., Kallweit, P., Mitra, N., Tiggel, B.S.,"Heat transfer enhancement and drag by longitudinal vortex generators in channel flow", J. Exp. Thermal Fluid Sci., Vol. 4, 1991, pp: 103-114.

[11] Douglas, J.F., Gasiorek, J.M., Swaffield, J.A.,"Fluid Mechanics", 3rd ed. Longman Scientific and Technical, 1996, pp: 300.

[12] Nasiruddin, M.H. Kamran Siddiqui,"Heat transfer augmentation in a heat exchanger tube using a baffle", International Journal of Heat and Fluid Flow, 2007, pp: 318-328.

[13] Sharma Sanjay Kumar, Sharma Vikas,"Maximizing the heat transfer through fins using CFD as a tool", International Journal of Recent advances in Mechanical Engineering, Vol. 2, No. 3, 2013.

[14] Demartini, L. C.,Vielmo, H. A, Moller, S. V,"Numerical and Experimental Analysis of the Turbulent Flow Through a Channel with Baffle Plates", Journal of the Braz. Soc. of Mech. Sci. and Eng., 2004.

[15] Gunter, A. Y., Sennestrom, H. R.,"A Study of Flow Patterns in Baffled Heat Exchangers", ASME Preprint, 47, 1947.