Numerical Study on Heat Transfer Enhancement in Heat Exchanger Tube Using Baffle

[1]Dr. Sourabh Khurana, [2]Pooja, [3]Kailash Prakash
[1]Dept. of Mechanical Engineering, Gurukul Vidyapeeth, Rajpura, Punjab, India
[2]Instructor of Ansys at Parametric Cad-Tech sector 8-C
[3]Parametric CadTech Sector 8-C Chandigarh, UT, India

Abstract
In heat exchanger tube heat transfer enhancement by baffle is reported. In the present investigation simulations were conducted on Ansys software version R 15.0 on a two dimensional domain, which represents a heat exchanger tube of 10 cm diameter and 100 cm length. Meshing has been done on ICEM CFD software. In the present paper rectangular and triangular two different shapes of baffles are considered for Reynolds number of 5000. The results shows that water flows from inlet to outlet in case of rectangular baffle is 2.5 m/s while in case of triangular baffle it is 2.1 m/s. the above result suggest that different shapes of baffles are considered so that heat transfer rate can be increased.

Keywords
Baffles, Ansys, ICEM, Reynolds number, Velocity, Vortices.

I. Introduction
High performance of heat exchangers is required in a wide range of engineering applications such as chemical engineering, power generation, automobile and aerospace industry [1]. The main objective of heat exchanger is to efficiently transfer of heat from one fluid to another separated by a solid wall. This can be achieved by introducing baffles inside the tube of heat exchanger with in line or in staggered arrays. This is because baffle helps to interrupt the hydrodynamic and thermal boundary layer and to induce the recirculation zone or vortices flow behind the baffle. This leads to increase heat transfer rate [2]. These vortices are produced by introducing an obstacle in a flow which is known as vortex generator. The effect of baffles on heat transfer rate, spacing between baffles and pressure drop have been studied by various investigators, but in the present study two different shapes of baffles are used and after that it was analyzed that which shape of baffle increases better heat transfer. This study has been conducted on Ansys CFD (Computational Fluid Dynamics) software version R 15.0 and meshing has been done on ICEM CFD software, so that fine mesh size can be obtained.

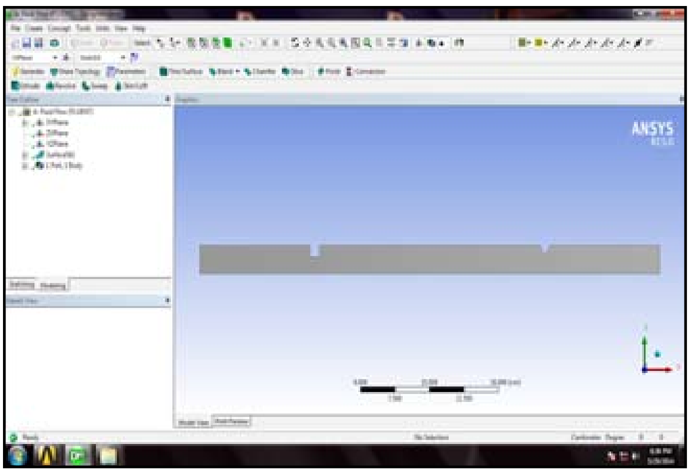


Fig. 1: Geometry of Rectangular Baffle

A. Numerical Method
The numerical simulations were conducted in a two dimensional domain, which represents a heat exchanger tube of 10 cm diameter and 100 cm length. The symmetry of geometry allows simulating half of the tube diameter using four boundary conditions which are inlet, wall, outlet and centerline. A baffle was used as a vortex generator, which was attached to the tube wall at a distance of 50 cm from upstream end of the tube. Two different baffles shapes were considered, the geometry of rectangular baffle is shown in fig. 1. The boundary conditions used is shown in fig. 2. The geometry of triangular shape baffle is shown in fig. 3 and the boundary condition used in this profile is shown in fig. 4. The basic parameters for baffles are given in Table 1. The most important turbulence method is K-epsilon model which is used in this study. Fluid used in this study is water and heat is added to water through wall. The temperature of water was set equal to 300 k at the inlet of the tube. A constant temperature of 375 K was applied on the entire wall as thermal boundary condition. CFD software Ansys version 15 was used to simulate the velocity contours and Temperature contours. To generate the required mesh ICEM CFD software was used. The residual error less than 10-6 was set as convergence criteria. The governing flow equations were discretized by finite volume method. The governing flow equations (i.e., continuity, momentum and energy equations) used to simulate the incompressible steady fluid flow and heat transfer in the computational domain are given as [3]

$$\frac{(\partial \rho')}{\partial t'} = \rho' \frac{(\partial V')}{\partial x'} - \rho' V' \frac{\partial (\ln A')V}{\partial x'} - V' \frac{\partial \rho'}{\partial x'}$$

$$\frac{\partial V'}{\partial t'} = -V' \frac{\partial V'}{\partial x'} - \frac{1}{\gamma}\left(\frac{\partial T'}{\partial x'} + \frac{T'}{\rho'}\frac{\partial \rho'}{\partial x'}\right)$$

$$\frac{\partial T'}{\partial t'} = -V' \frac{\partial T'}{\partial x'} - \gamma - 1T'\left(\frac{\partial V'}{\partial x'} + V' \frac{\partial (\ln A)}{\partial x'}\right)$$

Table 1: Basic Parameters for Baffles

S.No.	Particulars	Details
1	Density	997 Kg/ m^3
2	Reynolds number	5000
3	Temperature at wall	375 K
4	Heat Exchanger Tube diameter	10 cm
5	Heat Exchanger Tube length	100 cm

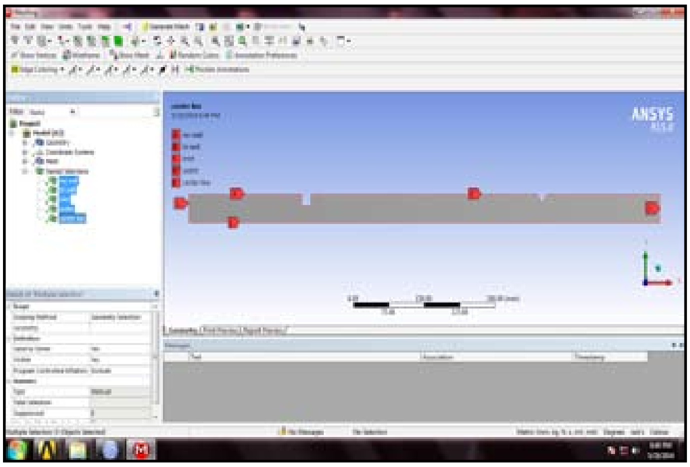


Fig. 2: Boundary Conditions for Rectangular Baffle

II. Literature Review

Berner et al. [4] suggested that a laminar behaviour for a channel with baffles is found at a Reynolds number below 600 and for such conditions the flow is free of vortex shedding. The numerical investigation of fluid flow and heat transfer characteristics in a smooth channel with staggered baffles.

Cheng and Huang [5] investigated the case of asymmetrical baffles and indicated that the friction factor shows a great dependence on baffle location, especially for a large height of baffle.

Cheng and Huang [6] again presented laminar forced convection in the entrance region of a horizontal channel with one or two pairs of baffles placed on the walls.

Fiebig et al. [7] experimentally investigated the effect of triangular and rectangular vortex generators on flow structures, flow losses and increase in heat transfer in compact heat exchanger.

Guo and Anand [8] studied the three dimensional heat transfer in a channel with a single baffle in the entrance region. Numerical studies for both solid and porous baffles in a two dimensional channel for the turbulent flow.

Mousavi and Hooman [9] numerically studied the heat transfer behavior in the entrance region of a channel with staggered baffles for eynolds numbers ranging from50 to 500 and baffle heights between 0 and 0.75 and reported that the Prandtl number affects the precise location of the periodically fully developed region.

Tsay et al. [10] numerically investigated enhancement of heat transfer by using baffles in laminar channel flow over two heated blocks mounted on the lower plate. The effect of baffle height, distance and thickness on flow structure was studied.

From the literature review it can be concluded that most of the studies was conducted by taking air as working fluid and even there is no such study conducted on different shapes of baffles in Ansys Software.

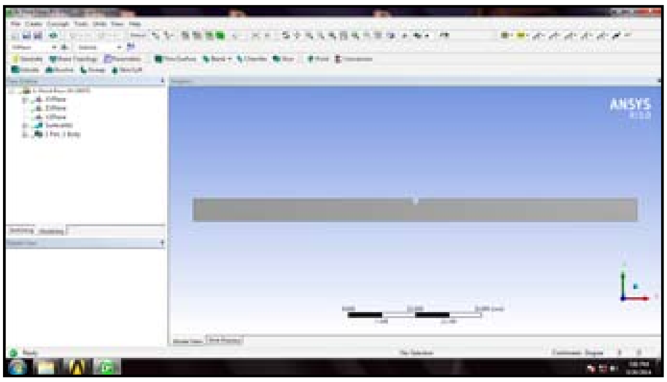


Fig. 3: Geometry of Triangular Baffle

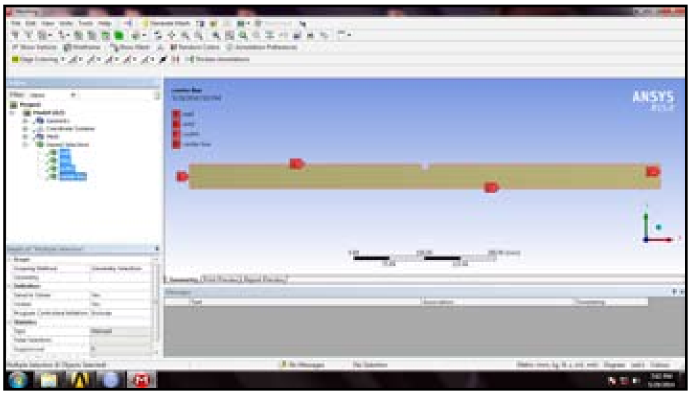


Fig. 4: Boundary Conditions for Triangular baffles

IV. Results and Discussion

In the present paper temperature contours and velocity contours have been developed for two different types of baffles shapes at Reynolds number of 5000. For this Ansys software has been used and simulation is done on tube of heat exchanger. Meshing for two different profiled has been done on ICEM CFD software which is shown in fig. 5 and 6. Four different types of boundary conditions were chosen for this task. In the present section we have presented the results obtained by Ansys software.

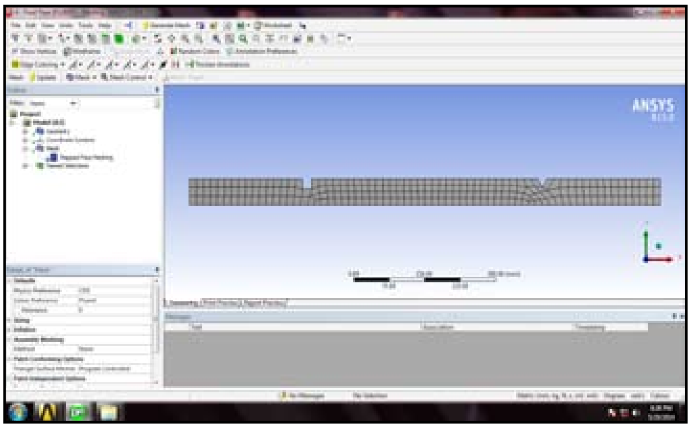


Fig. 5: Meshing for Rectangular Baffle

A. Velocity Vector

Fig.7 shows the velocity vectors for rectangular baffles. The Fig. shows the streamlines produced for rectangular baffle for Reynolds number = 5000. In this case a strong vortex is observed along the baffle. The red marks shows that this is the maximum velocity vector which is 2.5 m/s. The nodes adjacent to the wall and baffle were excluded since the high vorticity at these nodes were due to wall shear. Fig. 8 shows the velocity vectors for triangular baffle. This Fig. shows that maximum velocity vector is 2.19 m/s.

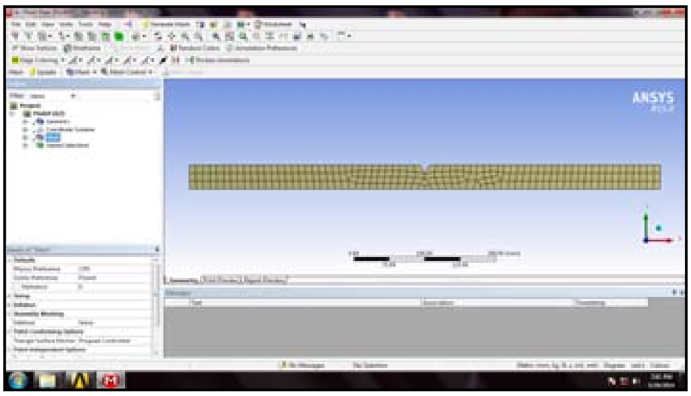


Fig. 6: Meshing for Triangular Baffle

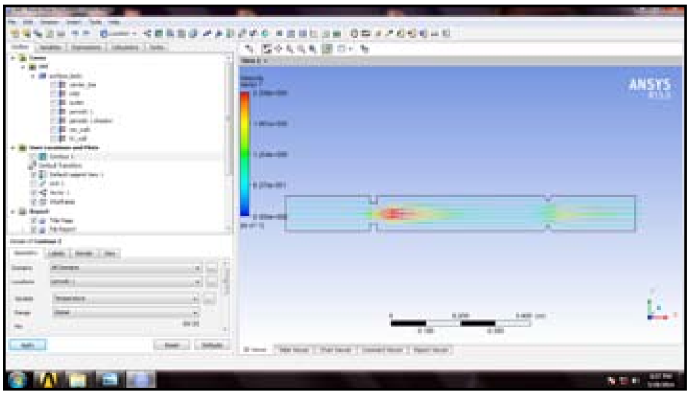


Fig. 7: Velocity vectors for rectangular baffle

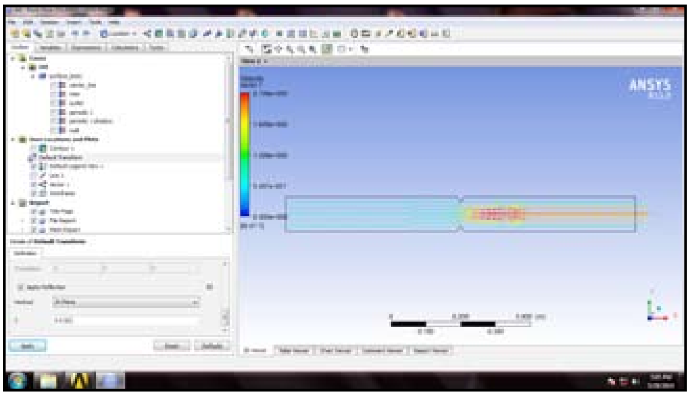


Fig. 8: Velocity vectors for triangular baffle

B. Temperature Contours

Fig. 9 and Fig. 10 show the temperature contours for rectangular and triangular baffles at Reynolds number = 5000, which shows a major change of temperature field along the tube, especially for the region opposite to baffle tip. This means that central vortex or recirculation zone provide a significant influence on the temperature field. That means the enhanced temperature region is mainly in the vortex influenced region.

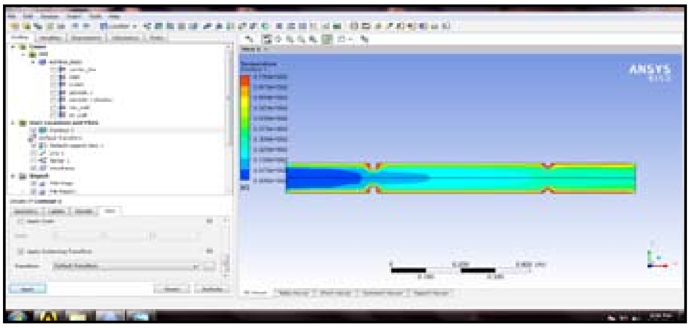


Fig. 9: Temperature Contours for Rectangular Baffles

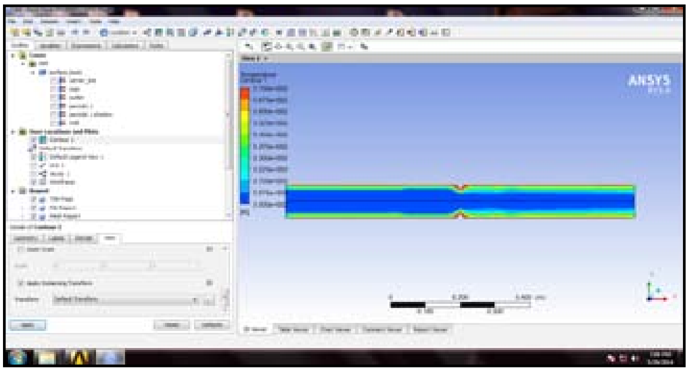


Fig. 10: Temperature contours of triangular baffles

V. Conclusion

Based on the numerical simulation which was conducted on Ansys software, it was concluded that when a baffle is introduced in the center of heat exchanger tube there is increase in temperature values and thus increases heat transfer. The effect of different types of shapes of baffles has been studied. The result shows that velocity vectors have high value for the case of rectangular baffles. The results also suggest that rectangular baffles increases the heat transfer rate than triangular baffles. Vortex influence the temperature field significantly.

VI. Scope for Future Work

The present work can be performed by experimental study and then this work can be compared to software.
Different type of shapes for baffles can be considered.

References

[1] M.H. Nasiruddin, Kamran Siddiqui,"Heat Transfer augmentation in a heat exchanger tube using a baffle", International Journal of Heat and fluid flow 28 (2007), pp. 318-328.

[2] Somchai Sripattanapipat, Pongjet Promvonge, "Numerical analysis of Laminar heat transfer in a channel with diamond based baffles", International Communications in heat and mass transfer 36 (2009) pp. 32-38.

[3] Sourabh Khurana, Anoop Kumar, Sourabh Kumar,"CFD Analysis of flow in a converging diverging nozzle using various turbulence models", International Conference on ICETME-2011 at Thapar University Patiala on 24-26 February 2011.

[4] C. Berner, F. Durst, D.M. McEligot,"Flow around Baffles", Trans. ASME J. Heat Transfer 106 (1984) pp. 743–749.

[5] C.H. Cheng, W.H. Huang, Laminar forced convection flows in horizontal channels with transverse fins placed in entrance regions, Int. J. Heat Mass Transfer 20 (1991) 1315–1324.

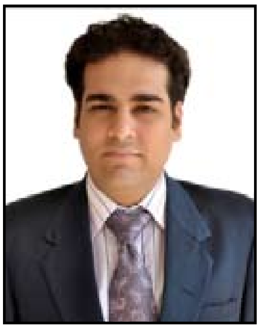
[6] C.H. Cheng, W.H. Huang,"Numerical prediction for laminar forced convection in parallel-plate channels with transverse fin arrays", Int. J. Heat Mass Transfer 34 (11) (1991) pp. 2739–2749.

[7] M.Fiebig, P. Kallweit, N. Mitra, "Heat Transfer enhancement and drag by longitudinal vortex generators in Channel flow", International Journal of Experimental Thermal and Fluid Sciences (4) (1991), pp. 103-114.

[8] Z. Guo, N.K. Anand,"Three-dimensional heat transfer in a channel with a baffle in the entrance region, Numer. Heat Transf., Appl. 31 (1) (1997) pp. 21–35.

[9] S.S. Mousavi, K. Hooman,"Heat and fluid flow in entrance region of a channel with staggered baffles, Energy Convers", Manag. 47 (2006), pp. 2011–2019.

[10] Y.L. Tsay, J.C. Cheng, T.S. Chang,"Enhancement of heat transfer from surface mounted block heat sources in a duct with baffles", Numer. Heat Transf., A Appl. 43 (8) (2003), pp. 827–841.



Dr. Sourabh Khurana has completed his B.Tech in 2005 in Mechanical Engineering from K.u.K. and then Joined M.Tech in 2006 at N.I.T. Hamirpur (H.P.) after completed M.Tech in Thermal, Author Joined Ph.D. at N.I.T Hamirpur (H.P.), He has completed his Ph.D. work related to hydro Turbines in April, 2014. He has published more than 20 Research papers in International Journals and 11 papers in International Conferences. Presently he is working as Associate Professor in Mechanical Engineering Department at Gurukul Vidyapeeth.

Mrs. Pooja is working as Instructor of Ansys at Parametric CadTech sector 8-C. She has completed her MS.C in 2006 and M.Phil in Environment science in 2007. She has published papers in Journals and Conferences.

Mr. Kailash Prakash runs Cad/ Cam Centre in Chandigargh sector 8-c, Parameteric CAD Tech centre. He is having very good knowledge of using Ansys software.