International Journal For Technological Research In Engineering (IJTRE)
Volume 2, Issue 7, March-2015
ISSN: 2347-4718

Effect of Reynolds Number on the Friction Characteristics of the Flow through a Heat Exchanger Tube with Baffles Using Ansys

Dr. Sourabh Khurana[1*], Khushmeet Kumar[2], Kailash Prakash[3]
[1]Associate Professor, Mechanical Engineering Department, GurukulVidyapeeth, Rajpura INDIA-140401
[2]Assistant Professor, Mechanical Engineering Department, Baddi University Baddi, INDIA-173205
[3]Parametric CadTech center sector 8-C Chandigarh INDIA-160008

Abstract **- In heat exchanger tube heat transfer enhancement by baffle which is at 45 degree is reported. In the present investigation simulations were conducted on Ansys software version R 14.5 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 shape of baffles are considered for different velocities. The results shows that water flows from inlet to outlet in case of rectangular baffle is 2.5 m/s. The above result suggest that rectangular shape of baffles are considered so that heat transfer rate can be increased.**

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

1. INTRODUCTION

High performance of heat exchangers is required in a wide range of engineering applications such as chemical engineering, power generation, automobile and aerospace industryNasiroddin and Siddiqui (2007). 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 rateSripattanapipat and Promvonge (2009). 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.

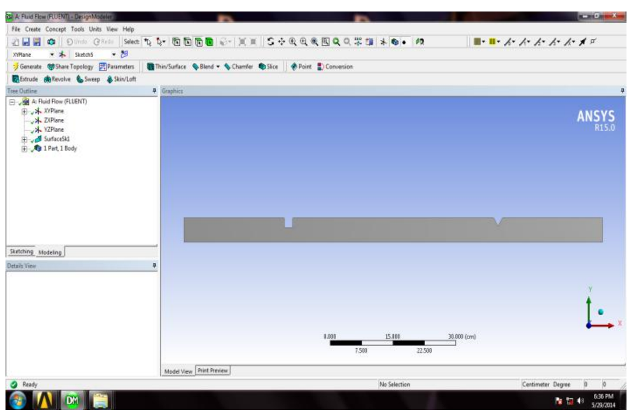

Figure 1: Geometry of Rectangular Baffle

1.1 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 Figure 1. The boundary conditions used are shown in Figure 2. The geometry of triangular shape baffle is shown in Figure 3 and the boundary condition used in this profile is shown in Figure 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 asKhurana et al (2011)

$$\frac{(\partial \rho^{'})}{\partial t^{'}} = \rho^{'} \frac{(\partial V^{'})}{\partial x^{'}} - \rho^{'} V^{'} \frac{\partial (\ln A^{'}) V}{\partial x^{'}} - V^{'} \frac{\partial \rho^{'}}{\partial x^{'}} \tag{1}$$

$$\frac{\partial V^{'}}{\partial t^{'}} = -V^{'} \frac{\partial V^{'}}{\partial x^{'}} - \frac{1}{\gamma}\left(\frac{\partial T^{'}}{\partial x^{'}} + \frac{T^{'}}{\rho^{'}} \frac{\partial \rho^{'}}{\partial x^{'}} \right) \tag{2}$$

$$\frac{\partial T^{'}}{\partial t^{'}} = -V^{'} \frac{\partial T^{'}}{\partial x^{'}} - \gamma - 1T^{'} \left(\frac{\partial V^{'}}{\partial x^{'}} + V^{'} \frac{\partial (\ln A^{'})}{\partial x^{'}} \right) \tag{3}$$

Table 1: Basic parameters for baffles

S.No.	Particulars	Details
1	Density	997 Kg/ m^3
2	Velocity	1.5, 2.5, 3.5 and 4.5
3	Temperature at wall	375 K
4	Heat Exchanger Tube diameter	10 cm
5	Heat Exchanger Tube length	100 cm

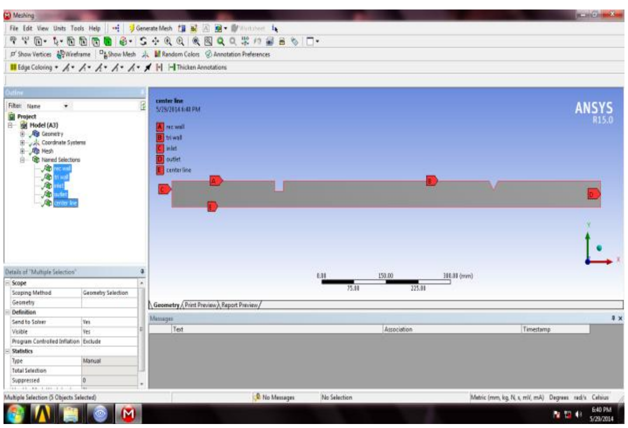

Figure 2: Boundary conditions for Rectangular Baffle

2. LITERATUREREVIEW

Berner et al.(1984) 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 (1991) 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 (1991) 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. (1991) 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(1997) 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(2006) numerically studied the heat transfer behavior in the entrance region of a channel with staggered baffles for Reynolds 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. (2006) 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.

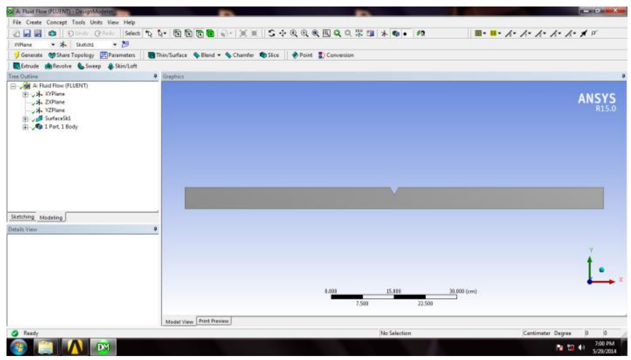

Figure 3: Geometry of Triangular Baffle

.

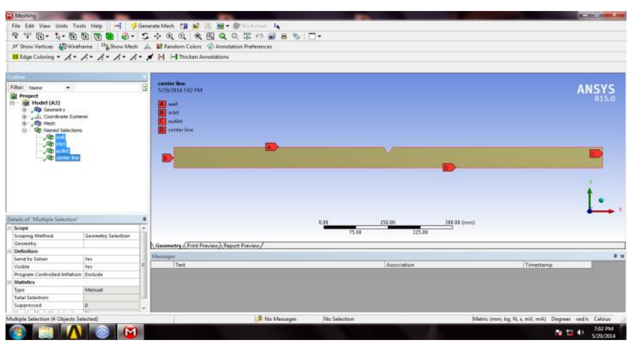

Figure 4: Boundary Conditions for Triangular baffles

3. RESULTS AND DISCUSSION

In the present paper temperature contours have been developed for rectangular baffles shapes for varying velocities of 1.5, 2.5, 3.5 and 4.5 m/s. For this Ansys software has been used and simulation is done on tube of heat exchanger. Meshing for profiles has been done on ICEM CFD software which is shown in Fig. 5 Four different types of boundary conditions were chosen for this task. In the present section we have presented the results obtained by Ansys software.

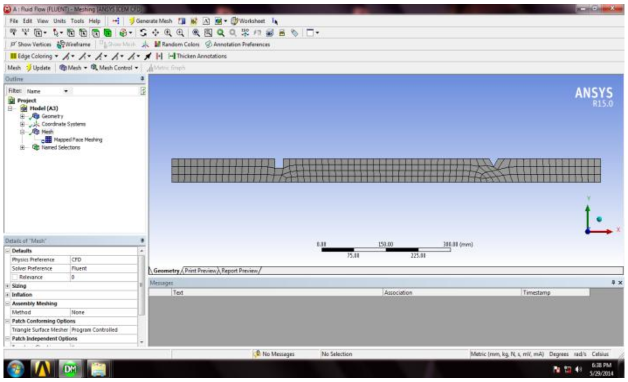

Figure 5: Meshing for Rectangular Baffle

3.1 Temperature contour

Fig.7 shows the velocity vectors for rectangular baffles. The Fig. shows the streamlines produced for rectangular baffle for velocity 1.5 m/s. In this case a strong vortex is observed along the baffle. The red marks shows that this is the maximum temperature which is 3.75×10^2 K. The nodes adjacent to the wall and baffle were excluded since the high vorticity at these nodes were due to wall shear. Fig. 7 shows the graph between temperature and distance and the temperature varies inside the tube and the results are plotted for centerline boundary conditon which shows the temperature is 309 K.

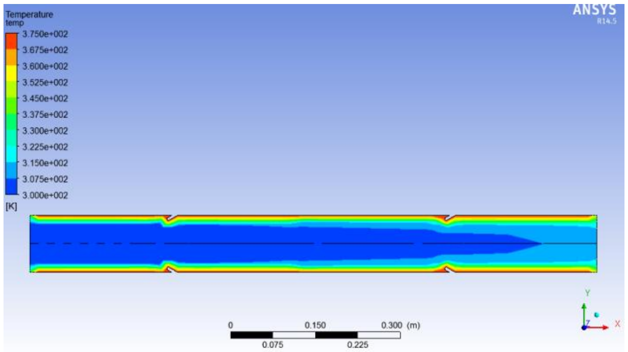

Figure 6: Temprature contour for rectangular baffle at 1.5 m/s

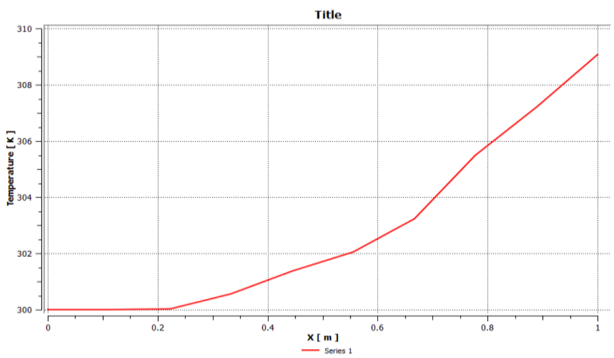

Figure 7: Temperture vs distance

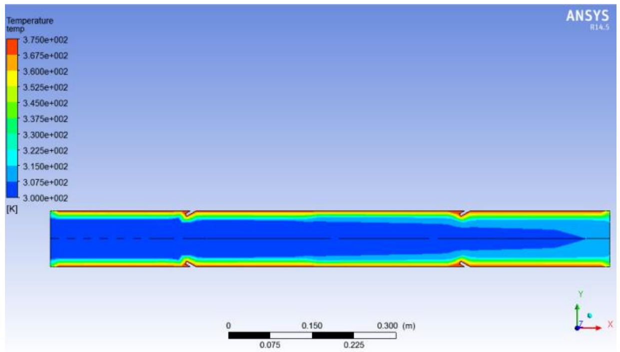

Figure 8: Temperature contours for rectangular baffle at 2.5 m/s

Fig. 8 show the temperature contours for rectangular baffles at velocity 2.5 m/s 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. While figure 9 shows that maximum value of temperature is 308.5 K.

Fig. 10 shows the teperature contours for 3.5 m/s velocity and maximum value for tempearture is 3.75×10^{2} K.. Fig. 11 shows the between temperature and distance which shows the maximum value of temperature 307.5 K.

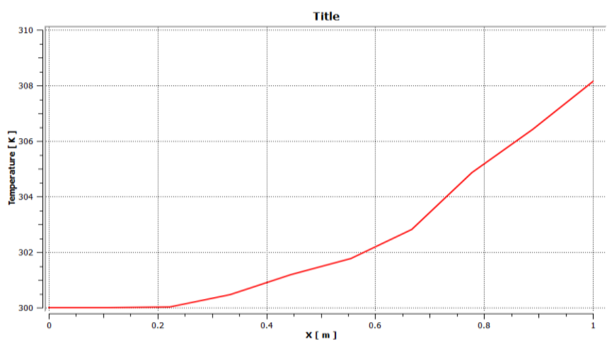

Figure 9: Temperture versus distance

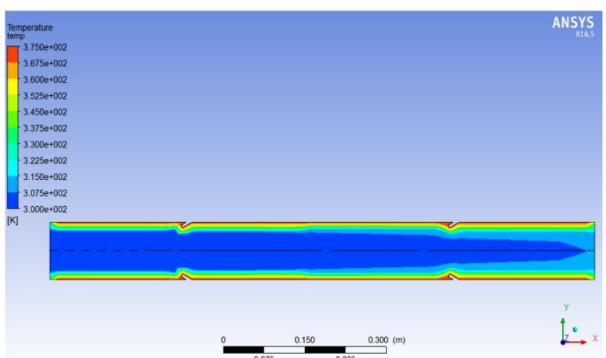

Figure 10: Temperature contours for rectangular baffles

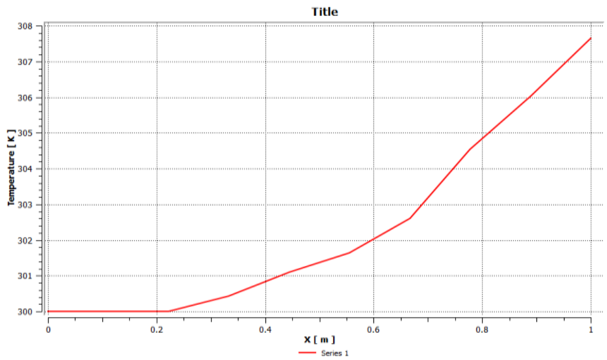

Figure 11: Temperature contours of triangular baffles

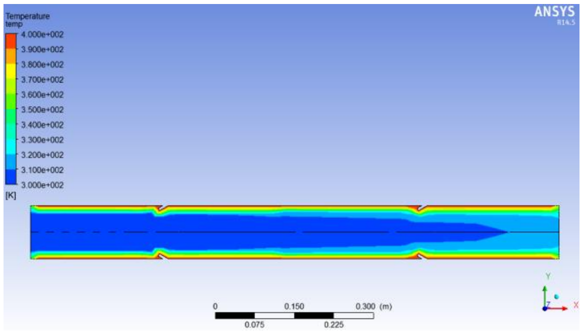

Figure 12: Temperature contours of triangular baffles

4. CONCLUSION

Based on the numerical simulation which was conducted on Ansys software, and temperture contours were plotted for radial and axial direction while the graph for distance were plotted for centerline. 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 rectangular shape of baffles has been studied. The result shows that temperature contours have high value for the case of 4.5 m/s for rectangular baffles. The results also suggest that rectangular baffles increases the heat transfer rate.

ACKNOWLEDGEMENT

This research is supported by Science and Engineering Research Council (SERC), Department of Science and Technology (DST) New Delhi, INDIA. (SR/SR3/MERC-098/2007)

REFERENCES

[1]. NasiruddinM.H. andSiddiquiK.(2007), Heat Transfer augmentation in a heat exchanger tube using a baffle. *International Journal of Heat and fluid flow*, Vol. 28 pp318-328.
[2]. Sripattanapipat S. and Promvonge P. (2009), Numerical analysis of laminar heat transfer in a channel with diamond based baffles. *International Communications in heat and mass transfer*,Vol. 36, pp 32-38.
[3]. Khurana S., Kumar A. andKumar S. (2011), 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]. BernerC., DurstF. and McEligotD.M. (1984), Flow around Baffles, *Trans. ASME J. Heat Transfer*,Vol. 106, pp743–749
[5]. ChengC.H. and HuangW.H. (1991), Laminar forced convection flows in horizontal channels with transverse fins placed in entrance regions, *Int. J. Heat Mass Transfer*,Vol. 20, pp 1315–1324.
[6]. ChengC.H. and HuangW.H. (1991), Numerical prediction for laminar forced convection in parallel-plate channels with transverse fin arrays, ,Vol. 34 N. 11pp 2739–2749.
[7]. FiebigM., KallweitP. andMitraN. (1991), Heat Transfer enhancement and drag by longitudinal vortex generators in Channel flow, *International Journal of Experimental Thermal and Fluid Sciences*,Vol.

4pp103-114.
[8]. GuoZ. andAnandN.K. (1997), Three-dimensional heat transfer in a channel with a baffle in the entrance region, *Numer.Heat Transf, Appl.*Vol. 31 N. 1pp 21–35.
[9]. MousaviS.S. andHoomanK. (2006), Heat and fluid flow in entrance region of a channel with staggered baffles, *Energy Convers. Manag.* Vol. 47 pp 2011–2019.
[10]. TsayY.L., ChengJ.C. and ChangT.S. (2003), Eenhancement of heat transfer from surface mounted block heat sources in a duct with baffles, *Numer. Heat Transf., A Appl.*Vol. 43 N. 8pp827–841.