ISSN: 2278-0181
Vol. 2 Issue 1, January- 2013

Cfd Analysis Of Heat Exchanger Over A Staggered Tube Bank For Different Angle Arrangement Of Tube Bundles

Karnav N Shah
Sardar Vallabhbhai Patel Institute of Technology, Vasad -388306, Gujarat, India

Kush V Mahant
Sardar Vallabhbhai Patel Institute of Technology, Vasad -388306, Gujarat, India

Priyank D Patel
Sardar Vallabhbhai Patel Institute of Technology, Vasad -388306, Gujarat, India

*Chetan O Yadav**
Assistant Professor
Sardar Vallabhbhai Patel Institute of Technology, Vasad -388306, Gujarat, India

Abstract

The modelling and performance prediction of a cross flow over a tube bundle using computational fluid dynamics (CFD) is the emerging development. The present work report, the analysis of pressure drop and heat transfer characteristics over a staggered tube bank heat exchanger with different tube bundle arrangements by using Fluent Software. The model was set up for a different mass flow rate over a tube bank and hence different Reynolds numbers and friction factor were studied. To improve hydraulic and thermal performance of heat exchanger we have simulated for the different angle arrangements i.e. 30°, 45° and 60°. The pressure drop results from the CFD simulation are compared with that obtained from the correlation.

1. Introduction

In scientific terms, the flow around circular cylinders includes a variety of fluid dynamics phenomena, such as separation, vortex shedding and the transition to turbulence. The mechanisms of vortex shedding and its suppression have significant effects on the various fluid-mechanical properties of practical interest: flow-induced forces such as drag and lift forces and pressure coefficient [1]. Furthermore, the predictions of turbulent cross-flow in a staggered tube bundle continue to attract interest due to its importance in the engineering application as well as the fact that this complex flow represents a challenging problem for CFD.

Cross-flow in tube bundles has wide practical applications in the design of heat exchangers, in flow across overhead cables, and in cooling systems for nuclear power plants [2]. In addition to the complexity arising from the flow instabilities in the tube bundle, one must also consider whether the flow is turbulent or laminar.

Due to symmetry in geometrical construction, a section of heat exchanger has been considered for CFD analysis by using fluent software. The Simple laminar flow model has been used to solve the transport equations for turbulent flow energy and the dissipation rate.

2. Computational Fluid Dynamics

The validation of the pressure drop results is obtained with the help of Correlations available from the friction factor and the developed results are compared with the CFD results. For finding the Reynolds number for the different mass flow rate ranging from 0.01 kg/s to 0.07 kg/s and thus, the friction factor are analyzed. The correlation equation for friction factor in terms of Reynolds number for pressure drop of a steady flow over a tube bundle:

Reynolds Number:

$$Re = \frac{\rho v D}{\mu} \quad \ldots eq^n. (1)$$

Friction Factor:

ISSN: 2278-0181
Vol. 2 Issue 1, January- 2013

$$f = \frac{16}{\text{Re}} \quad \ldots\text{eq}^{n}.\ (2)$$

Prandtl Number:

$$Pr = \frac{\mu C_p}{k} \quad \ldots\text{eq}^{n}.(3)$$

Nusselt Number:

$$Nu = 0.911 Pr^{0.33}\,\text{Re}^{0.385},\ 4 \le \text{Re} \le 40 \quad \ldots\text{eq}^{n}(4)$$

$$Nu = 0.683 Pr^{0.33}\,\text{Re}^{0.466},\ 40 < \text{Re} \le 4000 \ldots\text{eq}^{n}.(5)$$

Convective Heat Transfer Co-efficient:

$$h = \frac{Nuk}{L} \quad \ldots\ \text{eq}^{n}.\ (6)$$

Heat Transfer (Q):

$$Q = hA\Delta T \quad \ldots\ \text{eq}^{n}.\ (7)$$

Pressure Drop (Co-relation):

$$\Delta P = \frac{L\rho f}{2D} v^2 \quad [3] \quad \ldots\text{eq}^{n}.\ (8)$$

Preliminary efforts were done until comparable results are obtained for similar simulation results published in fluent. The problem is to simulate, a periodic flow and heat transfer over a 2D domain of a tube bank. The bank consists of uniformly spaced tubes with a diameter of 1 cm that is staggered in the direction of cross-fluid flow. Their centres are separated by a distance of 2 cm in the x direction, and 1 cm in the y direction. The bank has a depth of 1 m. Because of the symmetry of the tube bank geometry, only a portion of the domain needs to be modelled. The computational domain is shown in outline in Figure 2(a).

A mass flow rate of 0.05 kg/s is applied to the inflow boundary of the periodic module. Temperature of the tube wall (T_{wall}) is 400 K and the bulk temperature of the cross-flow water (T_1) is 300 K. The properties of water that are used in the model are shown in Figure 2(a).

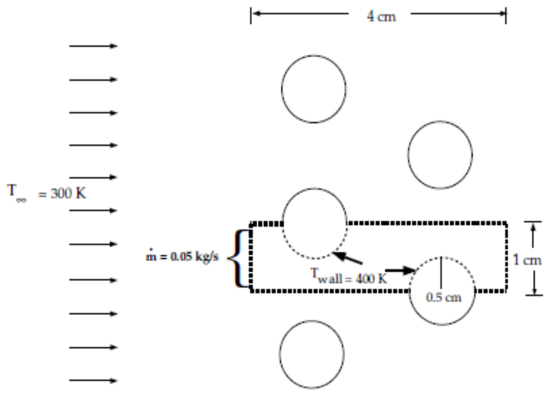

Figure.2 (a) Schematic diagram of the Problem [4]

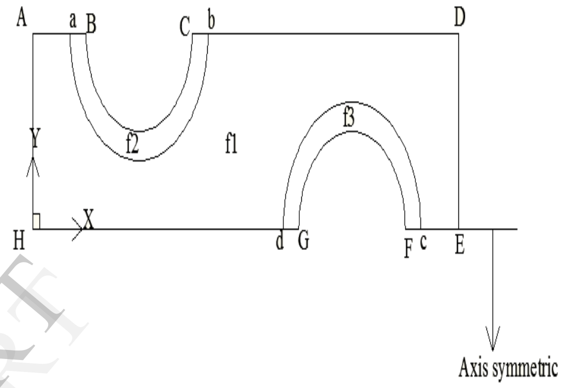

Figure.2 (b) Test section Model without Meshing

Table 1: Boundary Condition

Sr. No.	Component/Region	Boundary Condition
1	Line AH(Mass flow Inlet)	Inlet(300K, 0.05 kg/s)
2	Line AB(symmetric)	Symmetric1 (Adiabatic)
3	Arc ab(Interior)	Interior1
4	Line BC(Wall)	Wall1(400 K)
5	Line CD(symmetric)	Symmetric3
6	Line DE(Pressure outlet)	Outlet
7	Line GH(symmetric)	Symmetric2
8	Line EF(symmetric)	Symmetric4
9	Arc cd(Interior)	Interior2
10	Region ABCDEFGH (Fluid zone)	fluid

Table 2: Material Properties

Properties	**Value**
Mass flow rate, $\dot{m}$	0.05 kg/s
Dynamic Viscosity, μ	0.001003 kg/m-s
Density of Water, ρ	998.2 kg/m^3
Specific Heat of Water, C_p	4182 kJ/kg-K
Thermal conductivity, k	0.6 W/m-K
Length, L	1 m
Cross Sectional Area, A	0.01 m^2
Diameter, D	0.01 m

2.1. Grid Independency Study

The grid independent study was carried out for section. Studies were performed ondifferent meshing scheme to determine how mesh discretization altered inlet pressure. The coarser, medium, and finer grids were created and simulations were done on each grid for various pressure drop.

Table 3: Grid Comparison

Grid	**Interval Size**	**Cell**	**Face**	**Nodes**
Coarse	0.005	1,60,510	2,45,039	84,530
Medium	0.006	2,18,183	3,32,942	1,14,760
Finer	0.007	3,14,424	4,79,901	3,30,494

There are mainly two criteria for selection of interval size:

1) Mesh generation time
2) Effect of meshing on Pressure drop

Considering the above criteria, the pressure drop changes by a small amount for different interval size (i.e., 0.005, 0.006, and 0.007 cm). For finer mesh, mesh generation time is comparatively more than that of the time taken for coarse mesh and medium mesh. For solving this problem, we have taken medium mesh (interval size 0.006 cm) for our research work. For each case, the test geometry was created using hybrid mesh cells. Compared to a much finer meshing with 1, 14, 760 nodes it was found that the medium mesh model performed much faster without significant loss in solution accuracy. Thus, medium mesh grid with 2, 18,183 cells is selected for the simulation.

3. Result and discussion

CFD simulation was conducted for a periodic flow and heat transfer over staggered tube bank 2D modelled in FLUENT. The symmetry geometry is selected for a model due to the periodic nature. The mass flow rate of the cross-flow is known, and the model is used to predict the behavior of fluid flow at different angle arrangement. Different meshing scheme were generated in the Gambit. The quadrilateral mesh is used around the tubes and triangles elsewhere i.e. hybrid mesh is used for the domain. Material properties, mass flow and physical models i.e. viscous are specified within the limits. The model was set up for a different mass flow rate over a tube bank and hence different Reynolds numbers are studied.

For 60° longitudinal arrangement various contours are shown in the figure 3 (a), 3 (b) and 3 (c).

Cross flow over a cylinder exhibits complex flow patterns. The fluid approaching the cylinder will branch out and encircle the cylinder, forming a boundary layer that wraps around the cylinder. The fluid particles on the mid-plane will strike the cylinder at the stagnation point, bringing the fluid to complete stop and thus raising the pressure at that point [5]. The pressure decreases in a flow direction while the fluid velocity increases.

At very low stream velocities the fluid completely wraps around the cylinder and the two arms of the fluid meet on the rear side of the cylinder in an ordinary manner. Thus the fluid follows the curvature of the cylinder.

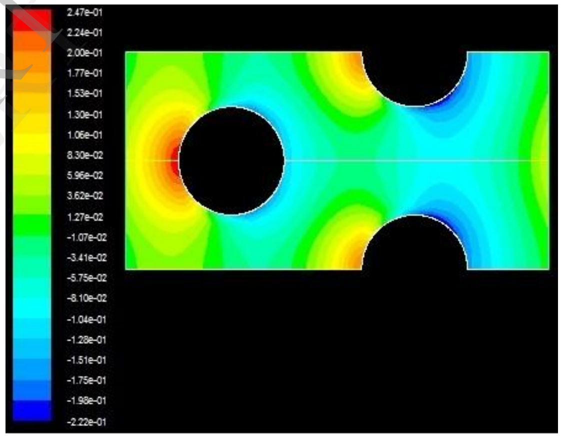

Figure.3 (a) Pressure Contour

At higher velocities, the fluid still hugs the cylinder on the frontal side, but it is to fast to remain attached to the surface as it approaches the top of the cylinder. As a result, the boundary layer detaches from the surface, forming a wake behind the cylinder. This point is called the separation point. Flow in the wake region is characterized by random vortex formation and pressures much lower than the stagnation point pressure.

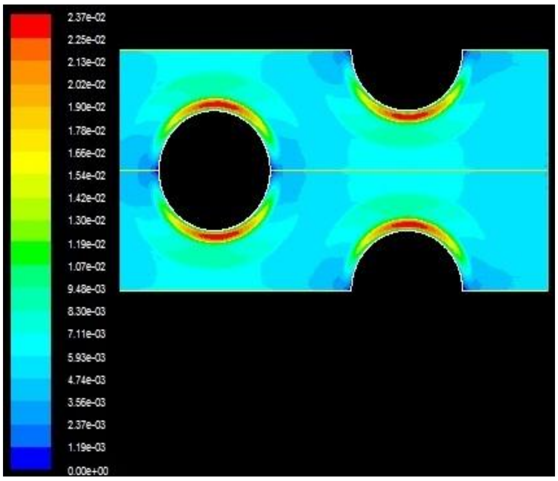

Figure.3 (b) Velocity Contour

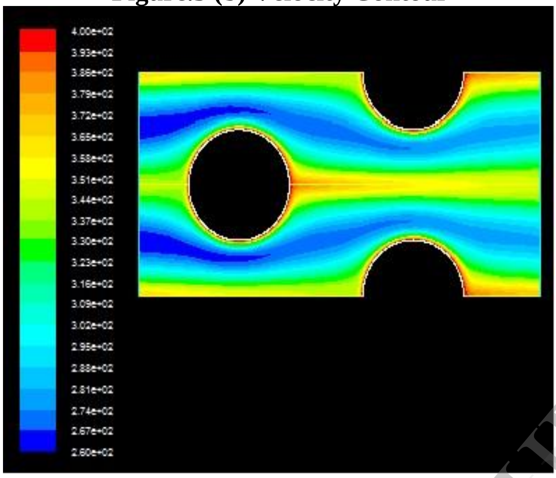

Figure.3 (c) Temperature Contour

Pressure and Temperature Contour for 45° longitudinal arrangement are shown in figure 3 (d) & 3 (e).

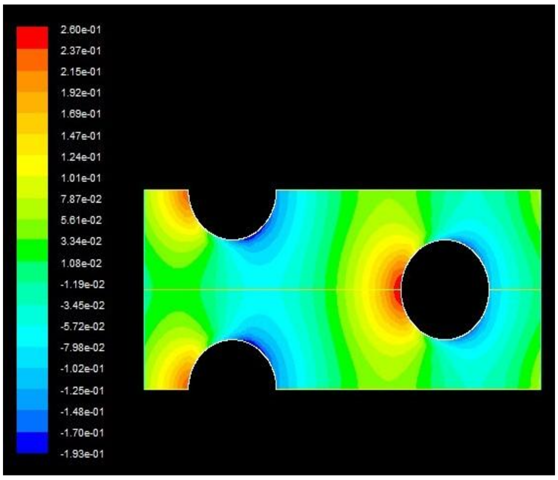

Figure.3 (d) Pressure Contour

In pressure contour, negative tendency behind the tube indicates separation of flow occurs at mostly upmost point of the tube. From that point, wake is generated behind the tube. The high pressure near the frontal area of tube indicates stagnation condition. From the above figure we can clearly observe that there is an early flow separation at 45° longudinal arrangement. The heat transfer near the surface region of the tube is more, compared to the other region because the surface of the tube is at higher temperature because of the hot fluid flowing inside the tube bank.

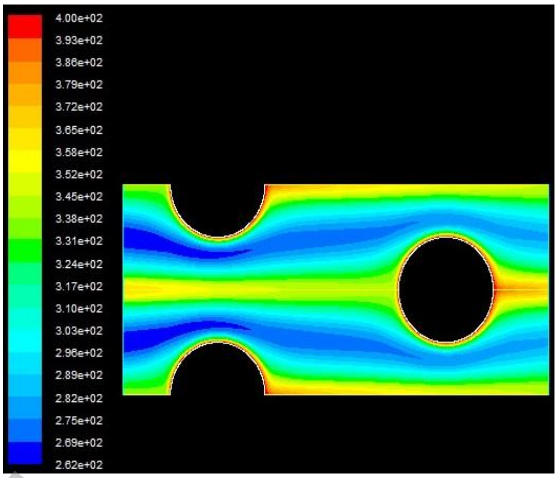

Figure.3 (e) Temperature Contour

Pressure drop increases linearly with respect to the mass flow rate. At 45° and 60° angle arrangement we have obtained a good agreement between correlation and Fluent result for different mass flow rate ranging from 0.01 kg/s to 0.07 kg/s are shown in figure 3(f).

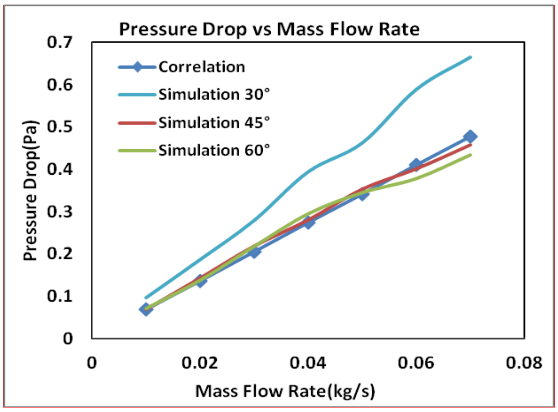

Figure.3 (f) Pressure drop as a function of mass flow rate

We observe that pressure drop v/s mass flow rate obtained from correlation follows the similar pattern for all angle arrangement when we changed only the longitudinal pitch but for the change in transverse pitch, pressure drop v/s mass flow rate obtained from correlation are different as seen from figure 3 (g).

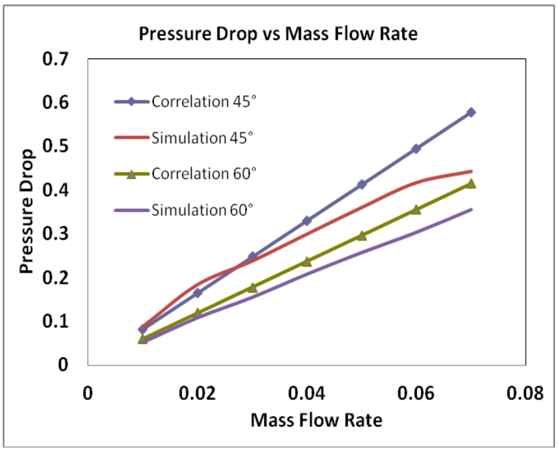

Figure.3 (g) Pressure drop as a function of mass flow rate

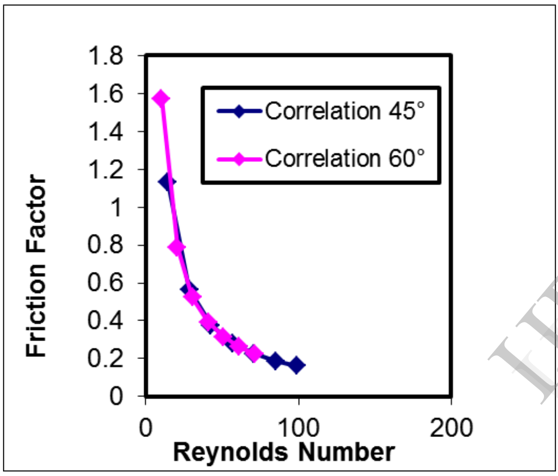

Figure.3 (h) Effect of Reynolds number on friction factor

Figure 3 (h) shows the friction factor over a common range of Reynolds number. Lower Reynolds number represents the effect of the growing linear viscous forces that tends to dominate the pressure drop while for a high Reynolds number, the quadratic Forchheimer's term tends to dominate.

4. Conclusion

Computational fluid dynamics (CFD) is the best tool for predicting fluid flow, heat & mass transfer, related phenomena by solving numerically the set of governing mathematical equations, conservation of mass, momentum, energy, species, etc. prior to the physical setup of the experiments. From the present work it is observed that when a periodic flow and heat transfer over a staggered tube bank for a different mass flow rate ranging from 0.01 kg/s to 0.07 kg/s, there is almost linear increment can be seen by the behavior of fluid flow parameters. The pressure drop results from the CFD simulation are compared with that obtained from the correlation.

Good agreement is observed for the pressure drop between simulation and correlation results for 45° (S_T = const.) angle arrangement as compare to the 60° (S_T = const.) angle arrangement, but the pressure drop obtained in 60° is less compare to the 45°.

5. References

1. A.Zukauskas and J. Ziugzda, "Heat Transfer of a Cylinder in Cross Flow", Washington D. C: Hemisphere, 1985.
2. T. Sreenivasulu*, BVSSS Prasad "Flow and Heat Transfer Characteristics in a Seven Tube-Bundle Wrapped with Helical Wires".
3. Yunus A. Cengel, "Heat Transfer- A Practical Approach".
4. Fluent 6.2 Inc. Help file tutorial 02.
5. Balabani, S., Bergeles, G., Burry, D., Yianneskis, M., (1994). "Velocity characteristics of the cross-flow over tube bundles", In: International Proceedings of the 7th International Symposium on Application of Laser Anemometry to Fluid Mechanics, Lisbon, vol. 2, p.39.3.1–39.3.8.