

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

CFD Analysis of Cross Flow Heat Exchanger and Experimental Validation

Utkarsh Wani¹, Shivam Kulkarni², Abhishek Keskar³

Design Analyst, Dept. of Mechanical Engineering, a² Innovation Research and Training, Warje, Pune, India¹

Design Analyst, Dept. of Mechanical Engineering, a² Innovation Research and Training, Warje, Pune, India²

Head Design Analyst, Dept. of Mechanical Engineering, a² Innovation Research and Training, Warje, Pune, India³

ABSTRACT: Enhancing the heat transfer by use of an elliptical shaped tube blank has been studied and research by many researcher because the fluid dynamics outside the elliptical shaped tube offers the more advantage over the circular shaped tubes. In present study, 2D-CFD analysis of the elliptical shaped tube in outside insulated cross flow heat exchanger with in-line arrangement and 45° angle of attack is carried out. The inlet and outlet temperature of hot water is constant at 80°C passed through an array of 30 SS204 tubes arranged at 45° angle of attack. The tubes are 150mm long with longitudinal and transverse pitch 52mm. The tubes are oriented at 45° angle of attack with direction of air flow. Thermal hydraulic performance of tube is carried out by varying mass flow rate of air and its effect on Nusselt number, Reynolds number, convective heat transfer coefficient, overall heat transfer coefficient is studied and validated with experimental results using ANSYS FLUENT 18.1 where the governing equations of mass, momentum and heat transfer were solved simultaneously, using the k-epsilon turbulence model. Result shows that various contours and vectors of temperature, velocity, pressure distributions, eddies formation and pressure drop at the end of tube is visualized. The percentage of error in outlet temperature of air from validation of CFD and experimental is 1 to 3 % and the enhancement ratio of elliptical shaped tube blank is 1.3 to 1.7 range compare to circular shaped tube blank in 9273.9 to 21000 Reynolds number range.

KEYWORDS: Cross flow heat exchanger, Elliptical tube blank, CFD Analysis

I. INTRODUCTION

A heat exchanger is a device that is used to transfer thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. A heat exchanger consists of heat transfer elements such as a core containing the heat transfer surface, and fluid distribution elements such as headers, manifolds, tanks, inlet and outlet nozzles or pipes, or seals. Usually, there are no moving parts in a heat exchanger; however, there are exceptions, such as a rotary regenerative exchanger in which the matrix is mechanically driven to rotate at some design speed or a scraped surface heat exchanger. The heat transfer is a surface of the exchanger core that is in direct contact with fluids and through which heat is transferred by conduction. That portion of the surface that is in direct contact with both the hot and cold fluids and transfers heat between them is transfer to as the primary or direct surface. To increase the heat transfer area, appendages may be intimately connected to the primary surface to provide an extended, secondary, or indirect surface.

Classification of Heat Exchanger based on Flow Arrangement

The choice of a particular flow arrangement is dependent on the required exchanger effectiveness, available pressure drops, minimum and maximum velocities allowed, fluid flow paths, allowable thermal stresses, temperature levels, piping and plumbing consideration, and other design criteria.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

a) Parallel flow exchanger:

In a parallel flow exchanger, the fluid streams enter together at one end, flow parallel to each other in the same direction, and leave together at the other end. This arrangement has the lowest exchanger effectiveness among single-pass exchangers for given overall thermal conductance (UA) and fluid flow rates (actually, fluid heat capacity rates) and fluid inlet temperatures; however, some multi-pass exchangers may have an even lower effectiveness, as discussed later. Fig. 1.1 shows the parallel flow heat exchanger.

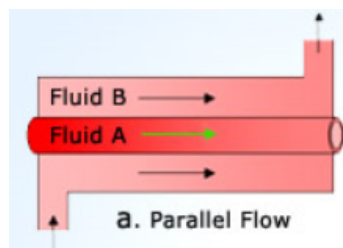


Fig. 1.1 Parallel Flow Heat Exchanger

b) Counter flow exchangers:

In a counter flow or counter current exchanger the two fluids flow parallel to each other but in opposite directions within the core. The counter flow arrangement is thermodynamically superior to any other flow arrangement. It is the most efficient flow arrangement, producing the highest temperature change in each fluid compared to any other two-fluid flow arrangements for a given overall thermal conductance (UA), fluid flow rates (actually, fluid heat capacity rates), and fluid inlet temperatures. Fig. 1.2 shows the counter flow heat exchanger.

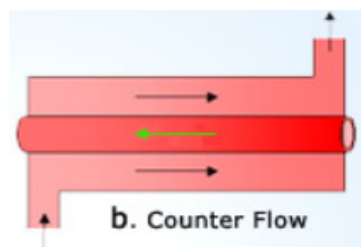


Fig. 1.2 Counter Flow Heat Exchanger

c) Cross flow exchanger:

In this type of exchanger the two fluids flow in directions normal to each other. Thermodynamically, the effectiveness for the cross flow exchanger falls in between that for the counter flow and parallel flow arrangements. This is one of the most common flow arrangements used for extended surface heat exchangers, because it greatly simplifies the header design at the entrance and exit of each fluid. Fig. 1.3 shows the cross flow heat exchanger.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

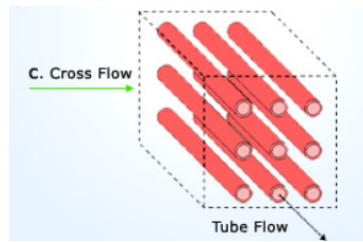


Fig. 1.3 Cross Flow Heat Exchanger

Heat Transfer Coefficient

Convective heat transfer, is the transfer of heat from one place to another by the movement of fluids due to the difference in density across a film of the surrounding fluid over the hot surface. Through this film heat transfer takes place by thermal conduction and as thermal conductivity of most fluids is low, the main resistance lies there. Heat transfer through the film can be enhanced by increasing the velocity of the fluid flowing over the surface which results in reduction in thickness of film. The equation for rate of heat transfer by convection under steady state is given by,

$$Q = h A (T_w - T_{atm})$$

Where, h is the film coefficient or surface coefficient (W/m²k)

A is the area of the wall,

T_w is the wall temperature and T_{atm} is surrounding temperature.

The value of h depends upon the properties of fluid within the film region; hence it is called 'Heat Transfer Coefficient'. It depends on the different properties of fluid, dimensions of the surface and velocity of the fluid flow (i.e. nature of flow).

The overall heat transfer coefficient is the overall transfer rate of a series or parallel combination of convective and conductive walls. The 'Overall Heat Transfer Coefficient' is expressed in terms of thermal resistances and its inverse is the overall heat transfer coefficient, U.

$$\frac{1}{U} = \frac{1}{h_o} + \frac{A_o}{A_i} \frac{1}{h_i} + R_{fo} + \frac{A_o}{A_i} R_{fi} + R_w$$

Where, U=Overall heat transfer coefficient based on outside area of tube all

A= area of tube wall

h= convective heat transfer coefficient

R_f= thermal resistance due to fouling

R_w = thermal resistance due to wall conduction and suffixes 'O' and 'I' refer to the outer and inner tubes, respectively.

Due to existence of the secondary flow, the heat transfer rates (& the fluid pressure drop) are greater in case of a curved tube than in corresponding straight tube at the same flow rate and the same temperature and same boundary condition.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Fig 1.3 shows the wall convection

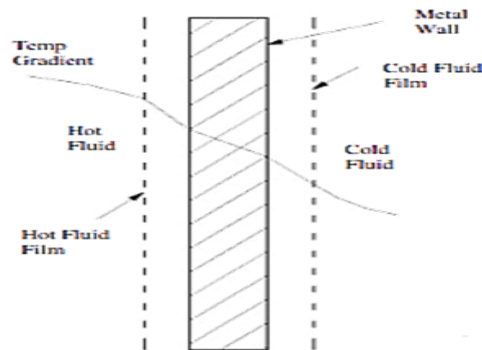


Fig.1.3 Wall Convection

Computational Fluid Dynamics

Computational Fluid Dynamics, abbreviated as CFD, uses different numerical methods and a number of computerized algorithms in order to solve and analyse problems that involve the flow of fluids. The calculations required to simulate the interaction of fluids with surfaces defined by boundary conditions, and initial conditions are done by the ANSYS Fluent v13.0. The Navier Stokes equations form the basis of all CFD problems. Two equation models are used for the simulations, and different models are discussed below.

The continuity equation, energy equation and the Navier-Stokes momentum equation govern the flow of the fluid in the curve tubes.

Continuity Equation gives the conservation of mass and is given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_1}{\partial x_1} + \frac{\partial \rho U_2}{\partial x_2} + \frac{\partial \rho U_3}{\partial x_3} = 0$$

$$\Rightarrow \frac{\partial U}{\partial x} + \frac{\partial V}{\partial x} = 0$$

And for constant density, $\frac{\partial \rho}{\partial t} = 0$

The momentum balance, (Navier-Stokes equations) follows Newton's 2nd law. The two forces acting on the finite element are the body and the surface forces. In CFD programmes, the momentum equation is given as

$$\rho \left(u \frac{\partial U}{\partial x} + v \frac{\partial V}{\partial x} \right) = -\rho g - \frac{\partial p}{\partial x} + \mu \frac{\partial^2 y}{\partial x^2}$$

The governing energy equation is given by:

$$\rho C_p \left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right) = k \frac{\partial^2 T}{\partial y^2}$$

Turbulence is created because of the unstable nature of the fluid flow. The flow becomes turbulent for higher Reynolds number. In this model the k-ε (turbulent kinetics energy "k" and the turbulent dissipation "ε") model is used.

The physical interpretation of the ε equation is,

1. Accumulation of ε
2. Convection of ε by the mean velocity
3. Production of ε

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

4. Dissipation of ε

5. Diffusion of ε

The time constant for turbulence is determined from the turbulent kinetic energy and dissipation rate of turbulent kinetic energy.

$$\tau = \frac{k}{\varepsilon}$$

Steps in CFD Analysis

Pre processor

- Geometry generation
- Geometry clean-up
- Meshing

Solver

- Problem specification
- Additional models
- Numerical computation

Post Processor

- Line and Contour data
- Average Value

II. LITREATURE REVIEW

The literature review is carried out in order to see the present research in this area which elaborated under present status. Toolthaisong et al. [1] conducted experimental investigation of the air side heat transfer and pressure drop characteristics at steady state for the cross flow heat exchangers having flat tubes. The results show that the thermal and pressure drop characteristics is governed by the angle of attack and axis ratio. The experimental setup consisted of flat tubes through which hot water was circulated. The air was passed over the tubes with velocity varying in the range of 2 to 6 m/sec. It was noticed that for all values of air velocity and axis ratio, the pressure drop and heat transfer rate increased with increasing angle of attack 0 to 90 degree; while the thermal hydraulic performance was decreased. For the tubes aspect ratio 0.18, 0.36 and 0.66, the pressure drop for 90 degree angle of attack was higher than 0 degree angle of attack by 17.8times, 5.5 times and 1.8 times respectively.

Ibrahim et al. [2] carried out experimental and numerical investigation of the turbulent flow through bundle of elliptic tubes heat exchanger. They investigated the effect of design parameters such as minor to major axis ratio and flow of angle of attack. The experiment was conducted on five bundle of elliptic tube heat exchanger with different axis ration. From experimental date it was found that increasing angle of attack clockwise until 90° enhances the convective heat transfer coefficient considerably. The heat exchanger with 0° angle of attack has the highest value of the efficiency index, which is the ratio between the Stanton number enhancement ratio (S_f/S_{f0}) to the friction factor ratio (f/f_0). The maximum thermal performance (heat transfer per unit pumping power) occurred at 0° whereas the minimum thermal performance of the elliptic occurred at 90° angle of attack. The elliptic tube bundle produced a considerable frontal area reduction of heat exchanger at 0° angle of attack with minor to major axis ratio 0.25 to 1.

Du et al. [3] worked on the two finned oval tubes heat exchanger (He1 : Double rows of tubes & He2 : Three rows of tubes). The heat exchangers were rotated about a pivot point so as to form different angles of attack with the direction of air flow. They investigated the effect of four inlet angles (30°, 45°, 60°, and 90°) on heat transfer and pressure drop performances of Reynolds number ranging from 1300 to 13000. The major axis to minor axis ratio of tubes was 2.57; arranged in staggered manner with longitudinal to transverse pitch ratio 2.25. Fin length to fin height ratio & fin eight

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

to fin spacing ratio was 2.16 and 22 respectively. They found out Nusselt number correlation for both heat exchangers with different angle of attack. In case of three row tubes heat exchanger with 45° angle of attack $Nu = 1.63 \times Re^{0.39}$ and $f = 70.79 Re^{-0.38}$ whereas for 90° angle of attack, $Nu = 2.09 \times Re^{0.37}$ & $f = 119 \times Re^{0.45}$ correlation was obtained by the researcher. The heat transfer rate for 30° , 45° , 60° angle of attack was compared with the heat transfer rate for 90° .

Sun et al. [4] compared the overall thermal performance of refrigerant to air finned tube condensers with elliptical and circular tubes. The result shows that the air pressure drop and the heat transfer rate of elliptical condenser are 20%-27.3% lower and 8.3% to 30.9% higher than those of circular tube condenser respectively. Compared to circular tube condenser, the system capacity improvement using elliptical condensers was found to be high a 21.3%-27.5% whereas the COP improvement rangers from 3.6to 6.7%. The study experiment was conducted using different refrigerants. The result shows the elliptical tube condenser using R410a yield higher improvement of condenser heat transfer rate and system capacity than using R22 and R134a.

Hui Han et al. [5] carried out experimental investigation on three different kinds of tubes with two types of enhanced fins, wavy fins and louvered fins. The results revealed that using the oval fin tube can not only reduce the flow resistance but also improve the host trader capacity go the heat exchangers. As comparing with the circle tube louver fin, the heat transfer rate of oval tube fin was found to be increased by 1.5-4.9% while the pressure drop loss was decreased by 22% to 31.8%.

Hasan et al. [6] analysed the performance of two evaporative cooled heat exchangers with plain circular tubes and other with oval tubes. It was concluded that the average mass transfer Colburn factor j for the oval tubes 89% of that for the circular tube, while the average friction factor for the oval tube is 46% of that for the circular tube. The ratio j/f for the oval tube is 1.93-1.96 times that for the circular tube. This means that for the oval tube has a better combined thermal-hydraulic performance.

Lavasani et al. [7] experimentally investigated flow around cam shaped tube bank with inline arrangement. It was found that friction factor of cam shaped tube bank is about 95% and 93% lower than the circular tube bank. By increasing longitudinally pitch ratios from 1.5to 2, heat transfer increases about 7%to 14%. Thermal hydraulic performance of cam shaped tube bank was about 6 times greater than circular tube bank due to aerodynamic shape of cam shaped tubes.

Khan et al. [8] experimentally investigated flows of hot air at $41.5 \pm 1.5^\circ C$ across an array of elliptical tubes carrying cold water. He results showed that the heat transfer rate increased with the increase in both water and air flows. The pressure coefficient (C_p), across a single tube in the array was found to remain at approx. 0.16 for $Re_a > 2 \times 10^4$. Both heat transfer rate and mussel number (Nu) were found to vary in power law relationship with Re . The Nu - Re correlation obtained from the idealized test bench was found to be in the form $Nu_a = 0.26 \times Re_a^{0.66}$.

Tao et al. [9] performed three dimensional numerical studies for laminar heat transfer and fluid flow characteristics of wavy fin heat exchangers with elliptic/circular tubes. A max relative heat transfer gain up to 30% was observed in the elliptic arrangement and corresponding friction factor only increased by about 10%. The result show that with the increasing of Reynolds number and fin thickness, decreasing of the eccentricity and span wise tube pitch, the heat transfer of the finned tube bank are enhanced with some penalty in pressure drop.

Zukauskaset al. [10] developed correlation for heat transfer and pressure drop for inline and staggered tube banks of Reynolds number from 1 to 2×10^6 and Prandtl number from 0.7 to 10^4 . A method optimizing the geometry of heat transfer surfaces is suggested by the researcher. The correlations developed by zakauskas are applicable for wide range of relative transfer and longitudinal pitches.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

III. CFD ANALYSIS

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, conclusions and discussions.

Assumption for CFD Analysis

1. In 2d analysis, we have consider flow visualization of air along length of tube blank is same.
2. The temperature of water flow through inside tube is constant at 80°C.
3. Conductance resistance is very low compare convective resistance hence neglected.
4. Outside surface of heat exchanger is insulated.

Geometry

Heat exchanger is built in CATIA V5R20. Heat exchanger is the cross flow heat exchanger. In that the water and air flow is perpendicular to each other.

First of all shell is created which is of rectangular shape having following dimensions:

- Length = 1000 mm
- Cross section dimension = (165 mm * 150 mm)
- Thickness = 1.25 mm

Further the shape of the elliptical tube pocket is done on the side wall of shell.

Total 30 tubes of 3 row and 10 column inline arrangement with longitudinal and transverse pitch of 52 mm are drawn. Then solid tubes with same dimensions with length 150 mm are created using extrude command. Finally assembled both the shell and tube. Table 3.1 shows the geometric dimensions of cross flow heat exchanger.

Table 3.1 Geometric Dimensions of Cross Flow Heat Exchanger

GEOMETRICAL DIMENSION OF HEAT EXCHANGER		VALUES(MM)
1	Length of shell (L)	1000
2	Width of shell (b)	148
3	Depth of shell(d)	163
4	Thickness of shell(t)	1.25
5	Major axis of elliptical tube	45.75
6	Minor axis of elliptical tube	23
7	Length of elliptical tube(L _B)	150.5
8	Angle of inclination	45 ^o
9	Longitudinal pitch	52
10	Transverse pitch	52
11	Thickness of tube(T _B)	1.25
12	Number of rows(N _R)	3
13	Number of column(N _C)	10
14	Number of tubes(N _T)	30

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Fig 3.1 shows the shell of cross flow heat exchanger.

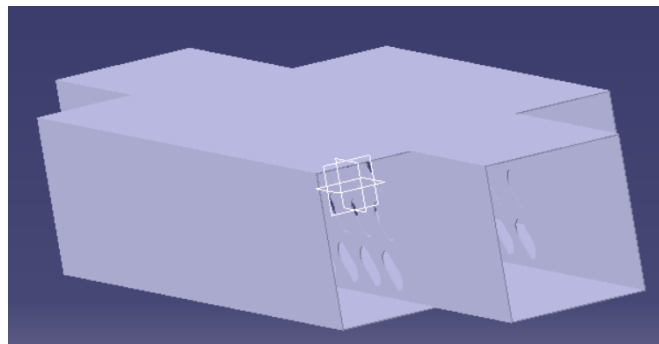


Figure 3.1 Shell of Cross Flow Heat Exchanger

Fig 3.2 shows the elliptical tube of cross flow heat exchanger.

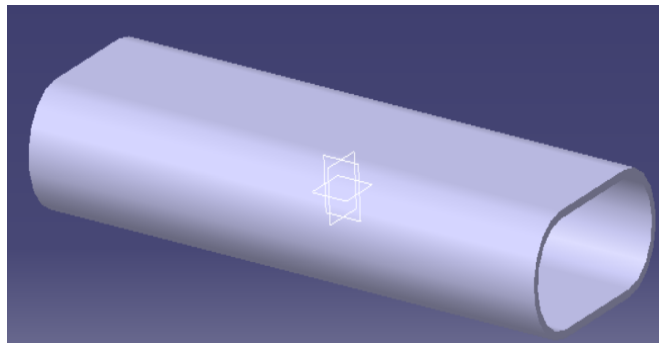


Figure 3.2 Elliptical Tube of Cross Flow Heat Exchanger.

Fig 3.3 shows the assembly of cross flow heat exchanger.

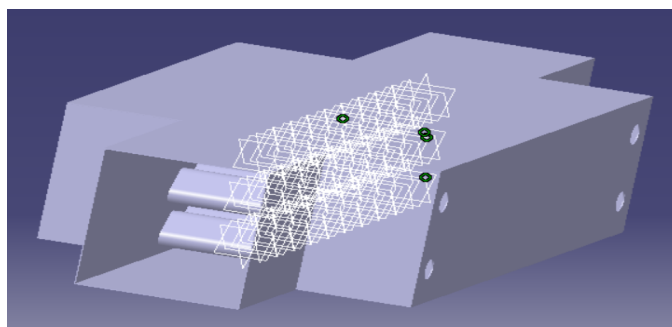


Figure 3.3 Assembly of Cross Flow Heat Exchanger

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Fig. 3.4 shows cross flow heat exchanger for 3D CFD simulation.

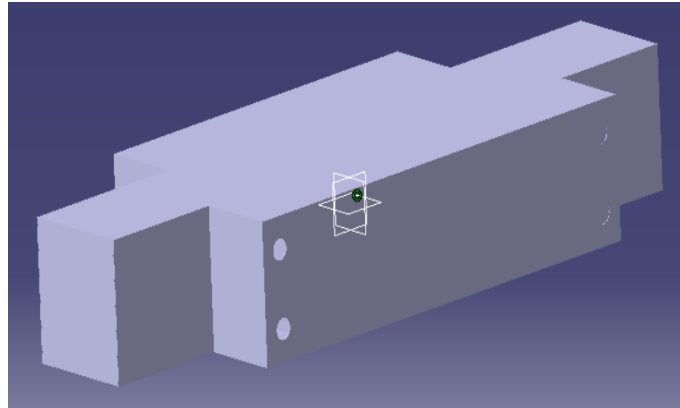


Figure 3.4 Cross Flow Heat Exchanger For 3D CFD Simulation

Mesh

Meshing of heat exchanger is done in ANSYS Workbench Design Module. It is cross flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modeller opens as a new window as the geometry is double clicked. Then the geometry which is in IGES format is imported from a stored file. Then double click on mesh, the meshing window opens.

Initially a relatively coarse mesh is generated. This mesh contains mixed cells having both triangular and quadrilateral elements. To improve the meshing we have applied the following steps:

i. Automatic method

Method – Quadrilateral Dominant
Element – First order (linear)
Free face mesh type – Quad/ Tri

ii. Sizing

Element size – 2 mm
Size function – uniform
Behaviour – soft
Growth rate – 1.2

iii. Inflation

Inflation option – smooth transition
Transition ratio – 0.5
Maximum layers – 6
Growth rate – 1.2
Boundary – Tube wall and Shell inner wall

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

iv. Named Selection

1. Inlet
2. Outlet
3. Boundary Wall
4. Tube Wall

After meshing the quality of meshing is checked. In that the following parameters related to quality of mesh were found:

- i. Skewness – 0.62902 (Max)
- ii. Aspect ratio – 5.2636 (Max)
- iii. Jacobian ratio – 4.7954 (Max)

From meshing number of nodes and elements are as follows:

Nodes – 46609
Elements – 45276

Fig. 3.5 shows 2D mesh of flow domain.

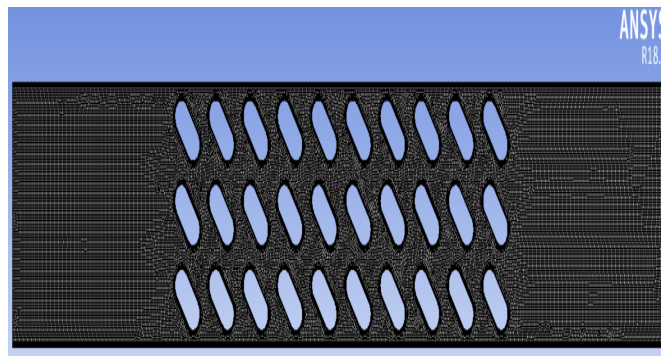


Figure 3.5 2D Mesh of Flow Domain

Fig. 3.6 shows mesh structure near tube wall.

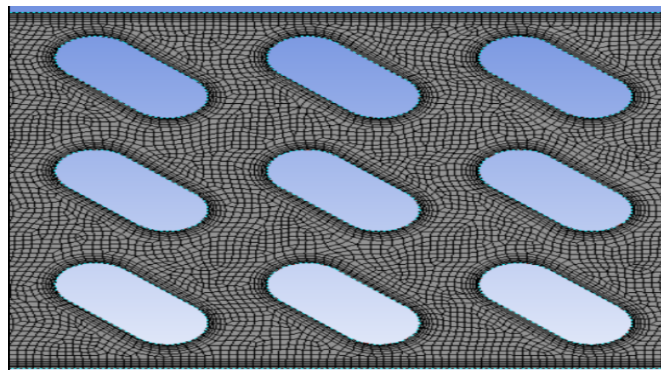


Figure 3.6 Mesh Structure Near Tube Wall

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Solution

After generation of meshing double click on setup new window of fluent solver opens.

➤ Problem Setup

The mesh is checked and quality is obtained. The analysis type is changed to the pressure based type. The velocity formulation is changed to the absolute and time to steady state. Gravity is defined as $y = -9.81 \text{ m/sec}^2$

➤ Models

Energy is set to on position. The flow is turbulent so realisable k-epsilon is selected in viscous model.

➤ Materials

Create / edit option is clicked to add air and steel to the list of fluid and solid respectively from fluent data base.

Air properties

Density – 1.225 kg/m^3
Specific Heat – 1006.23 J/kgK
Thermal Conductivity – 0.0242 W/mk
Viscosity – $1.7894 \times 10^{-5} \text{ kg/m-sec}$

Steel properties

Density – 8030 kg/m^3
Specific Heat – 502.48 J/kgK
Thermal Conductivity – 16.27 W/mk

➤ Cell Zone Condition

The part are assigned as air and steel as per fluid/ solid part.

➤ Boundary Condition

Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as mass flow inlet and pressure outlet. The tube wall and shell wall are separately specified with boundary condition. The details about all boundary conditions can be seen in given below. Table 3.2 shows the boundary conditions.

Table 3.2 Boundary conditions

Boundary	Condition	Value
Inlet condition of Air	Mass flow inlet	0.06 kg/sec
	Temp. inlet	315.3 k
Outlet condition of Air	Pressure outlet	0 Pa
Tube	No slip	
	Constant temperature	353 k
Inner shell wall	No slip	
	Heat flux	0

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

➤ Reference values

The inner inlet is selected from drop down list of “Compute from”. The values are:

- Area – 1 m²
- Density – 1.225 kg/m³
- Depth – 1 m
- Enthalpy – 100125.6 J/kg
- Length – 1 m
- Pressure – 0 Pa
- Temperature – 315.3 k
- Velocity – 0.3004883 m/s
- Viscosity – 1.7894e-5 kg/m-sec
- Ratio of specific heat – 1.4

➤ Solution Method

The solution methods are specified as follows:

- Scheme - simple
- Gradient – least square cell base
- Pressure – second order
- Momentum – second order upwind
- Turbulent kinetic energy – 1st order upwind
- Turbulent dissipation rate – 1st order upwind
- Energy – 2nd order upwind

➤ Solution Control and Initialization

Under relaxation factors the parameters are

- Pressure – 0.3 Pa
- Density – 1 kg/m³
- Body forces – 1 kg/m²sec²
- Momentum – 0.7 kg-m/sec
- Turbulent kinetic energy – 0.8 m²/s²
- Turbulent dissipation rate – 0.8 m²/s³

Then the solution initialization method is set to hybrid initialization.

➤ Run Calculation

The number of iteration is set to 750 and the solution is calculated and various contours, vectors and plots are obtained. Fig 3.7 shows the iteration solution graph.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

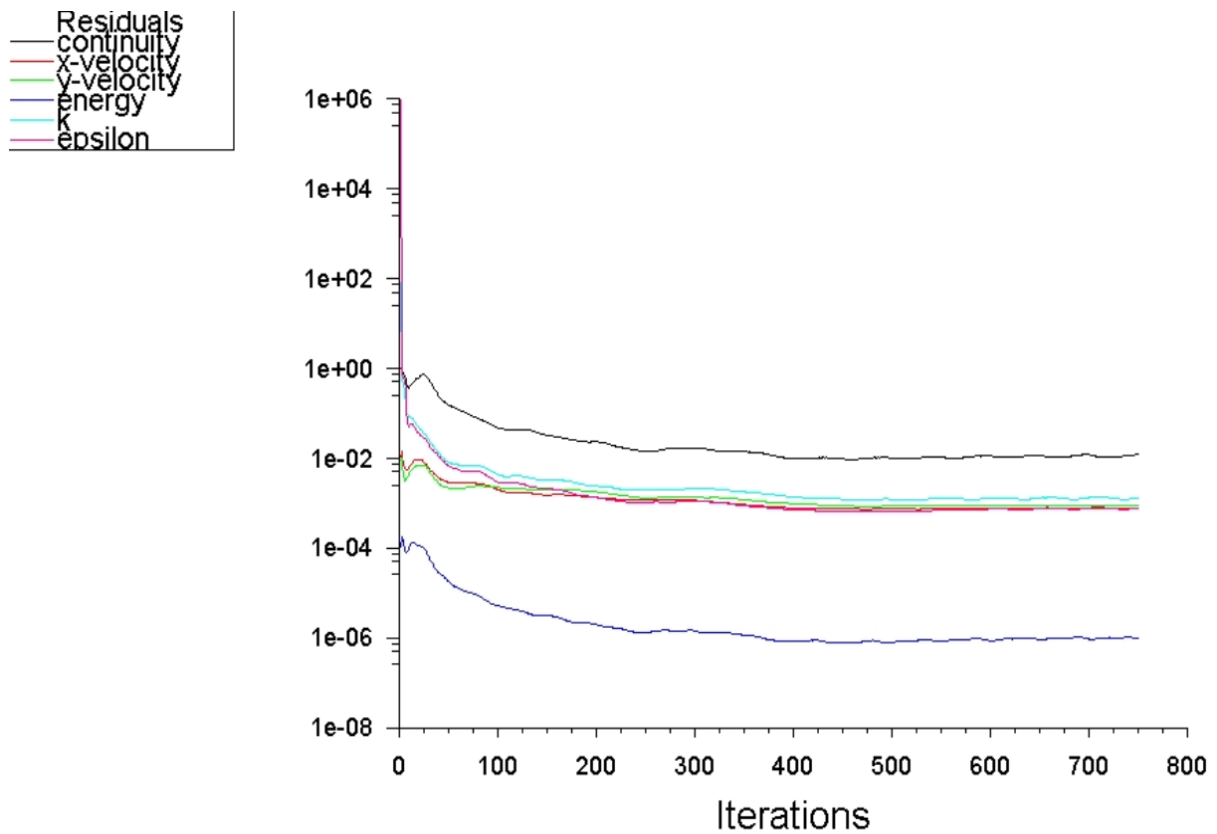


Figure 3.7 Iteration solution graph

IV. RESULTS AND DISCUSSIONS

In results and discussion various contours, vectors, plots related to temperature, pressure, velocity and results of simulation are discussed.

Results from CFD

The results are obtained from Ansys fluent 18.1 by Simulation of cross flow heat exchanger for mass flow rate 0.06 kg/sec.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

➤ Contours

Fig 4.1 shows the temperature distribution in Kelvin

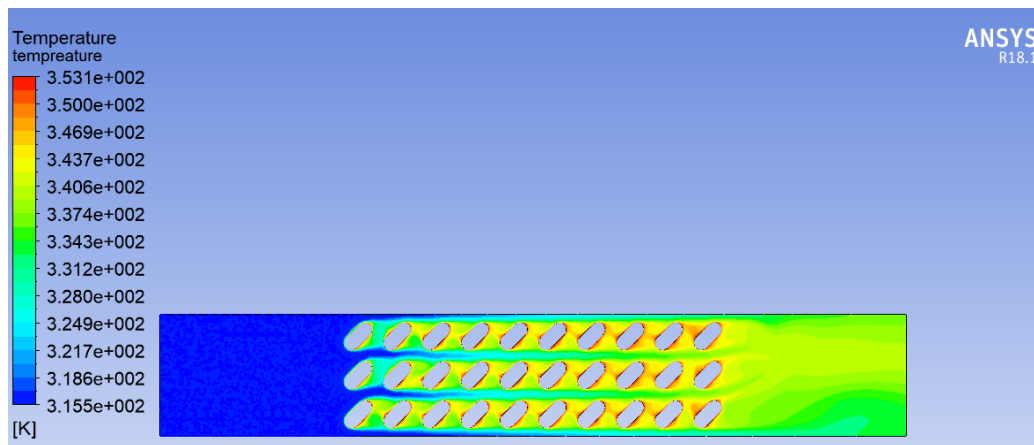


Figure 4.1 Contours of temperature distribution in K

Fig 4.2 shows the velocity distribution in m/sec

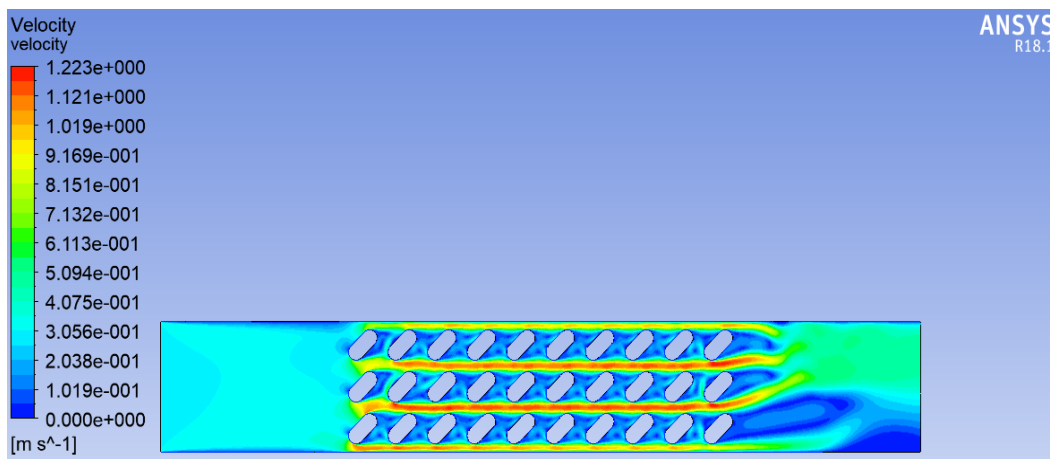


Figure 4.2 Contours of velocity distribution in m/sec

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Fig 4.3 shows pressure distribution in Pascal

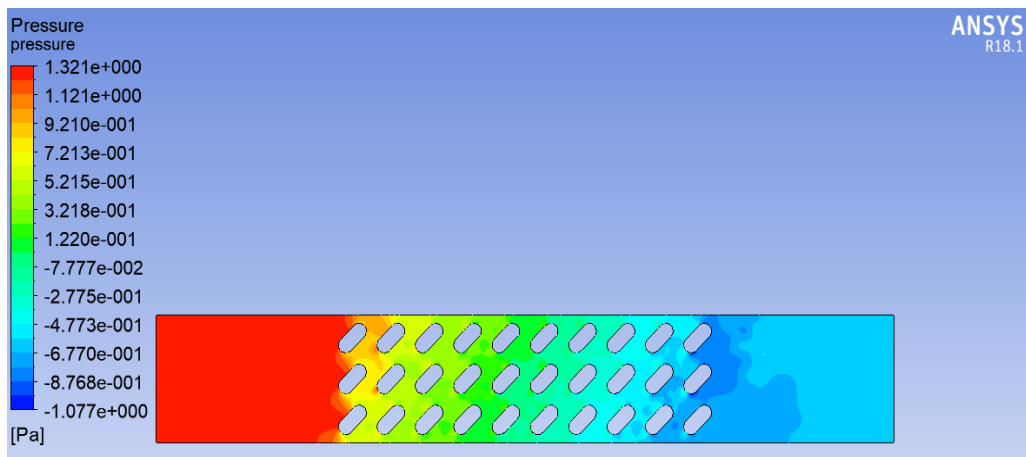


Figure 4.3 Contours of pressure distribution in Pascal

➤ Vectors

Fig. 4.4 shows the velocity vector distribution (m/sec)

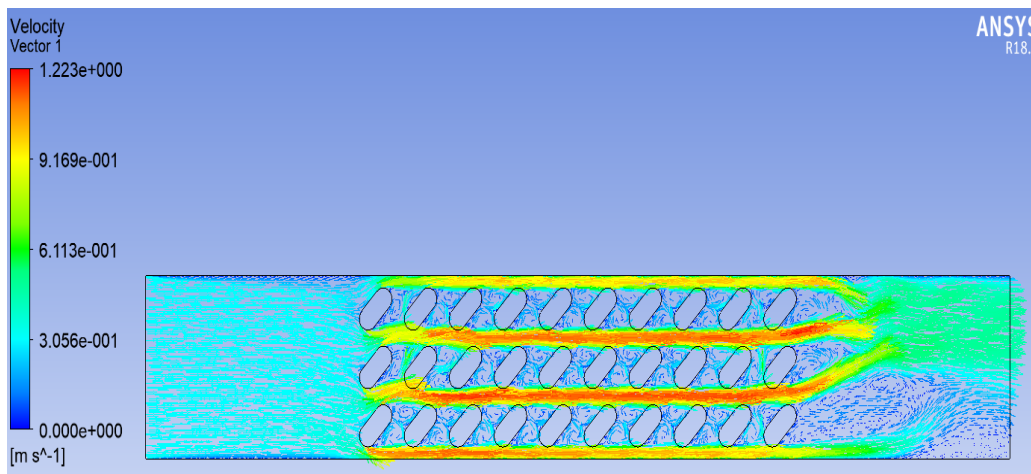


Fig 4.4 Velocity Vector Distribution (m/sec)

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Fig. 4.5 shows close view of eddies formation by velocity vector

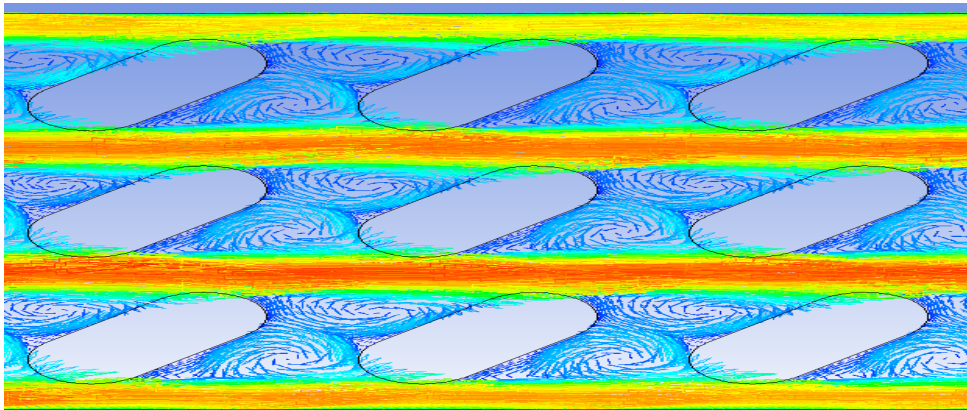


Figure 4.5 Close view of eddies formation by velocity vector

➤ Plots

Fig. 4.6 shows the variation of Reynold's Number with the temperature difference of air for various cases.

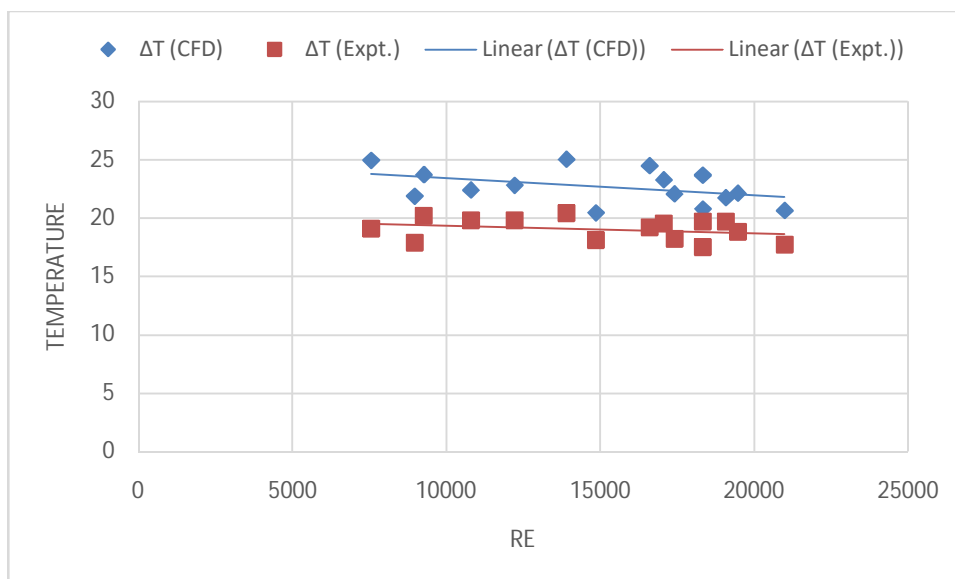


Figure 4.6 Reynolds number vs temperature difference of air

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Fig 4.7 shows variation of Reynold's Number with heat transfer coefficient for various cases.

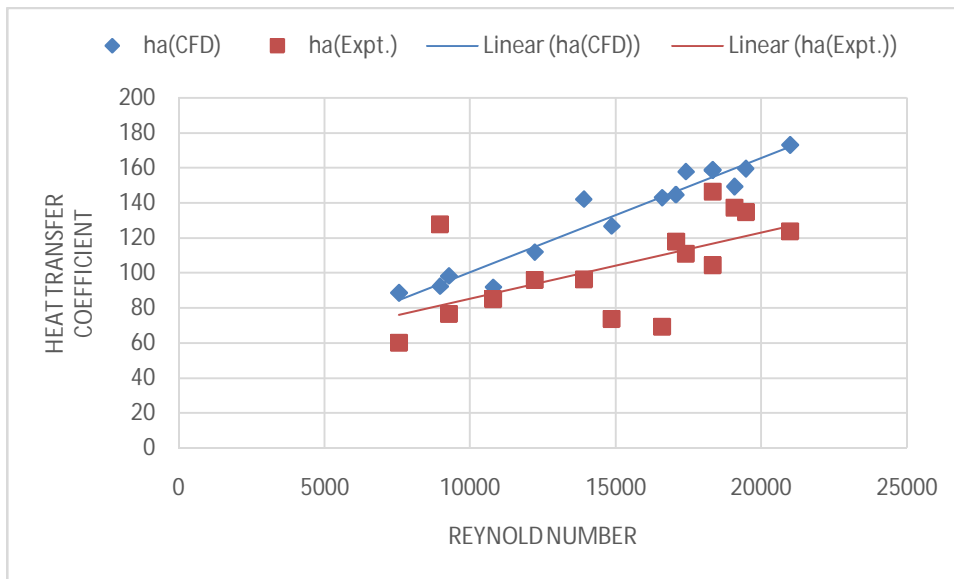


Figure 4.7 Reynolds Number vs Heat Transfer Coefficient

Fig 4.8 shows graph of variation of Reynold's Number vs Nusselt Number

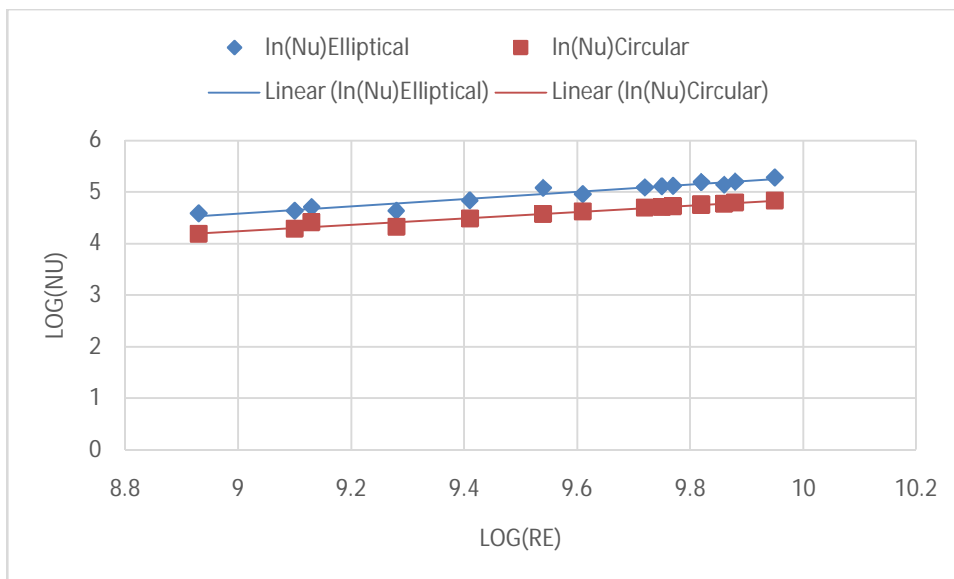


Figure 4.8 Reynolds Number vs Nusselt Number

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

CFD Calculations

Sample Calculation for reading 1

Water side calculation

Given data

Flow rate of water $(M_w)_{\text{constant}} = 1400$ LPH

Water inlet temperature $T_1 = 80^\circ\text{C}$

Water outlet temperature $T_2 = 80^\circ\text{C}$

Average mean temperature of water at inlet and outlet = 80°C

Properties at 80°C

Density of water $\rho = 971.8$ kg/m³

Thermal conductivity of water $K = 0.67$ W/m⁰k

Dynamic viscosity of water $\mu = 3.55 \times 10^{-4}$ N-sec/m²

Prandtl Number of water $Pr = 2.22$

$H_1 = 0.025$ m

$V_2 = 0.02075$ m

Formula

DittusBoelter Equation

$$Nu_w = 0.023 Re^{0.8} Pr^{0.3} \quad (1)$$

$$\text{Hydraulic diameter } (D_h)_w = \frac{4A}{P} \quad (2)$$

$$\text{Three Tube c/s area } A_{c/s(\text{tube})} = 3 \left(\frac{\pi}{4} \times (V_2)^2 + V_2 \times H_1 \right) \quad (3)$$

Velocity of water

$$(V)_w = \frac{m}{\rho \times A} \quad (4)$$

Reynolds Number of water

$$(Re)_w = \frac{\rho V D_h}{\mu} \quad (5)$$

Heat transfer coefficient of water

$$h_w = \frac{Nu \times K}{D_h} \quad (6)$$

Calculations

$$(D_h)_w = 0.029 \text{ m} \quad \dots\text{From (2)}$$

$$A_{c/s(\text{tube})} = 2.58 \times 10^{-3} \text{ m}^2 \quad \dots\text{From (3)}$$

$$(V)_w = 0.155 \text{ m/sec} \quad \dots\text{From (4)}$$

$$(Re)_w = 1.23 \times 10^4 \quad \dots\text{From (5)}$$

$$Nu_w = 54.8 \quad \dots\text{From (1)}$$

$$h_w = 1270 \text{ W/m}^2\text{k} \quad \dots\text{From (6)}$$

Air Side Calculations

Given data

Mass flow rate of air, $M_a = 0.06$ kg/sec

Inlet temperature of air $T_3 = 315.3$ °k

Outlet temperature of air $T_4 = 339.022$ °k

Properties of air at mean temperature

Specific heat of air $(C_p)_{\text{air}} = 1.00808$ kJ/kg-k

Number of tubes $N_T = 30$

$V_3 = 0.023$ m, $V_6 = 0.163$ m, $H_4 = 0.150$ m,

Maximum axis length of elliptical tube $G = 0.045$ m

Length of shell surface contact with air $F = 0.650$ m

Properties at T_m

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Density of Air	$\rho_{air} = 1.085 \text{ kg/m}^3$
Specific heat of air	$(C_p)_{air} = 1.0080 \text{ kJ/kg-k}$
Dynamic viscosity of air	$\mu_{air} = 202.76 \times 10^{-7} \text{ N-sec/m}^2$
Kinematic viscosity	$\nu_{air} = 19.762 \times 10^{-6} \text{ m}^2/\text{sec}$
Thermal conductivity of air	$K_{air} = 28.31 \times 10^{-3} \text{ W/m}^0\text{k}$
Prandtl Number	$P_r = 0.70319$

Formula

$$\text{Tube Length Surface Area } (A_{ts}) = (\pi \times V_3 + 2 \times H_1) \times H_4 \times N_T \quad (7)$$

$$\text{Side plate surface area } (A_{ps}) = 2 [V_6 \times F - (\frac{\pi}{4} \times V_3^2 + V_3 \times H_1) \times N_T] \quad (8)$$

$$\text{Average temperature, } T_m = \frac{T_3 + T_4}{2} \quad (9)$$

$$\text{Heat transfer area} = \text{Tube length surface area} + \text{Side plate surface area} \\ (A_s) = (A_{ts}) + (A_{ps}) \quad (10)$$

Heat absorbed by Air (Q)

$$Q = M_a C_p (T_4 - T_3) \quad (11)$$

Log Mean Temperature Difference (LMTD)

$$\text{LMTD} = \frac{(T_1 - T_4) - (T_2 - T_3)}{\ln \left(\frac{T_1 - T_4}{T_2 - T_3} \right)} \quad (12)$$

Overall heat transfer coefficient (U)

$$U = \frac{Q}{A \times \text{LMTD}} \quad (13)$$

Heat transfer coefficient of Air (h_a)

$$\frac{1}{U} = \frac{1}{h_a} + \frac{1}{h_w} \quad (14)$$

Nusselt number of Air for Elliptical tube (Nu)

$$(N_u)_{\text{elliptical}} = \frac{h_a \times (D_h)_{air}}{k} \quad (15)$$

$$\text{Velocity of air} = \frac{M}{\rho \times A_{min}} \quad (16)$$

Minimum area for air flow

$$A_{min} = H_4 \times [V_6 - (3 \sin 45^\circ \times G)] \quad (17)$$

Reynold number for air

$$(R_e)_{air} = \frac{\rho \nu D_{ah}}{\mu} \quad (18)$$

Zukauskas Equation

$$(N_u)_{\text{circular}} = 0.27 (R_e)^{0.63} (P_r)^{0.36} \quad (19)$$

Heat transfer coefficient of air from circular correlation

$$(h)_{\text{circular}} = \frac{N_u \times K}{D_{ah}} \quad (20)$$

$$\text{Enhancement Ratio} = \frac{h_{\text{elliptical}}}{h_{\text{circular}}} \quad (21)$$

Calculations

$$T_m = 327.161 \text{ k} \quad \dots \text{From (9)}$$

$$(A_s) = (A_{ts}) + (A_{ps}) \quad \dots \text{From (10)}$$

$$= 0.55016 + 0.15247$$

$$= 0.7026 \text{ m}^2$$

$$Q = 1434.699 \text{ W} \quad \dots \text{From (11)}$$

$$\text{LMTD} = 23.909 \quad \dots \text{From (12)}$$

$$U = 85.406 \text{ w/m}^2\text{k} \quad \dots \text{From (13)}$$

$$(h_a)_{\text{elliptical}} = 91.56 \text{ w/m}^2\text{k} \quad \dots \text{From (14)}$$

$$(N_u)_{\text{elliptical}} = 103.49 \quad \dots \text{From (15)}$$

$$A_{min} = 0.0099 \text{ m}^2 \quad \dots \text{From (17)}$$

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

(V) _{air} = 5.349 m/secFrom(16)
(R _e) _{air} = 9273.95From(18)
(N _u) _{circular} = 75.1080From(19)
(h) _{circular} = 65.628 w/m ² kFrom(20)
Enhancement Ratio = 1.395From (21)

Result of CFD Analysis and Experimental Investigation

Table 4.1 shows the variation of Mass flow rate, Inlet Temperature of Water, Outlet Temperature of Water, Inlet Temperature of Air, Outlet Temperature of Air, Overall heat transfer coefficient and Heat transfer coefficient of elliptical and circular tube from CFD Analysis.

Table 4.1 Observations and Results from CFD Analysis

Sr. No.	m _a kg/sec	T1 (°k)	T2 (°k)	T3 (°k)	T4 (°k)	U W/m ² k	(h _a) _{ellip.} W/m ² k	(h _a) _{cir.} W/m ² k
1	0.06	353	353	315.3	339.0	85.4	91.56	65.62
2	0.069	353	353	315.7	338.1	90.95	97.97	72.17
3	0.078	353	353	315	337.82	102.70	111.74	77.94
4	0.088	353	353	316.4	336.3	127.68	141.95	84.32
5	0.108	353	353	311.9	335.20	129.68	144.41	95.49
6	0.119	353	353	312.9	334.61	113.46	149.14	102.52
7	0.123	353	353	312.9	335.03	141.62	159.39	105.14
8	0.049	353	353	316.9	341.85	82.69	88.40	55.08
9	0.095	353	353	317.1	337.57	115.10	126.57	88.37
10	0.111	353	353	315.3	337.37	140.25	157.66	98.58
11	0.128	353	353	315.9	336.56	152.19	172.82	110.9
12	0.117	353	353	316.4	337.20	141.04	158.68	101.97
13	0.058	353	353	319.0	340.88	85.91	92.14	64.9
14	0.105	353	353	310	334.9	128.39	142.83	124.1
15	0.116	353	353	311.5	335.19	140.82	158.46	101.07

Table 4.2 shows Nusselt Number values for elliptical tube, circular tube, Reynolds number in air, ratio of heat transfer coefficient of elliptical tube and circular tube, Prandtl Number in air, Reynold's Number in Water, Nusselt Number in Water and Prandtl Number in Water by performing CFD analysis.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Table 4.2 Observations and Results from CFD Analysis

Sr. No.	(Nu) _{ellip.}	(Nu) _{cir.}	(Re) _a	$\frac{h_{ellip.}}{h_{cir.}}$	(Pr) _a	(Re) _w	(Nu) _w	(Pr) _w
1	103.49	75.1	9273.9	1.395	0.7032	12304.90	54.8	2.22
2	110.8	82.66	10796.8	1.35	0.7032	12304.90	54.8	2.22
3	126.6	89.39	12225.5	1.51	0.7033	12304.90	54.8	2.22
4	161.24	96.97	13909.4	1.68	0.0734	12304.90	54.8	2.22
5	164.78	110.34	17069.84	1.512	0.7037	12304.90	54.8	2.22
6	170.09	118.38	19087.66	1.45	0.7036	12304.90	54.8	2.22
7	181.71	119.86	19468.91	1.51	0.7036	12304.90	54.8	2.22
8	99.23	66.01	7558.61	1.52	0.7028	12304.90	54.8	2.22
9	143.01	101.1	14864.5	1.432	0.7031	12304.90	54.8	2.22
10	167.45	111.71	17414.2	1.59	0.7033	12304.90	54.8	2.22
11	195.94	125.67	20993.65	1.559	0.7033	12304.90	54.8	2.22
12	179.54	115.38	18333.66	1.55	0.7032	12304.90	54.8	2.22
13	103.41	73.54	8973.03	1.41	0.7028	12304.90	54.8	2.22
14	163.17	108.44	16605.4	1.317	0.7038	12304.90	54.8	2.22
15	180.90	115.41	18331.1	1.567	0.7037	12304.90	54.8	2.22

Table 4.3 shows the variation of Mass flow rate, Inlet Temperature of Water, Outlet Temperature of Water, Inlet Temperature of Air, Outlet Temperature of Air, Overall heat transfer coefficient and Heat transfer coefficient of elliptical and circular tube from Experimental Results.

Table 4.3 Observations and Results from Experimental Investigation

Sr. No.	m _a kg/sec	T1 (°k)	T2 (°k)	T3 (°k)	T4 (°k)	(h _a) _{ellip.} W/m ² k	(h _a) _{cir.} W/m ² k	(Nu) _{cir.}
1	0.06	352.6	351.7	315.3	335.5	76.48	68.27	76.68
2	0.069	353.1	352	315.7	335.5	85.01	72.02	80.84
3	0.078	352.9	351.9	315	334.8	95.74	78.15	87.90
4	0.088	353.4	352	311.3	331.7	96.35	83.21	94.45
5	0.108	352.7	351.6	311.9	331.4	117.82	94.92	107.71
6	0.119	353	352	312.9	332.6	137.07	101.31	114.61
7	0.123	352.6	351.4	312.9	331.7	134.46	103.26	116.95
8	0.049	352.5	352	316.9	336	60.08	58.24	65.24
9	0.095	353.9	352.2	317.1	335.5	110.99	88.33	98.98
10	0.111	353.4	353.4	316.2	334.4	123.62	97.25	109.26
11	0.128	352.5	351.2	315.9	333.6	146.28	106.67	120.02
12	0.117	353.3	352.3	316.4	333.9	127.59	100.81	113.31
13	0.058	353.8	351.4	319	336.9	69.24	64.87	72.37
14	0.105	352.9	352.1	310.3	329.5	104.44	93.11	106.15
15	0.116	352.8	351.2	311.5	331.2	127.97	99.23	112.68

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

Table 4.4 shows Nusselt Number values for elliptical tube, circular tube, Reynolds number in air, ratio of heat transfer coefficient of elliptical tube and circular tube, Prandtl Number in air, Reynold's Number in Water, Nusselt Number in Water and Prandtl Number in Water using Experimental Investigation.

Table 4.4 Observations and Results from Experimental Investigation

Sr. No.	m_a kg/sec	$(Re)_a$	$\frac{h_{ellip.}}{h_{cir.}}$	$(Pr)_a$	$(Re)_w$	$(Pr)_w$	$(Nu)_w$
1	0.06	9611.85	1.12	0.70	12345.09	2.22	54.83
2	0.069	10946.64	1.18	0.65	12345.09	2.22	54.83
3	0.078	12518.22	1.22	0.64	12345.09	2.22	54.83
4	0.088	14109.64	1.16	0.64	12345.09	2.22	54.83
5	0.108	17374.98	1.24	0.64	12345.09	2.22	54.83
6	0.119	19142.05	1.35	0.64	12345.09	2.22	54.83
7	0.123	19780.43	1.30	0.64	12345.09	2.22	54.83
8	0.049	7778.18	1.03	0.65	12345.09	2.22	54.83
9	0.095	15077.98	1.26	0.65	12345.09	2.22	54.83
10	0.111	17668.52	1.27	0.64	12345.09	2.22	54.83
11	0.128	20527.48	1.37	0.64	12345.09	2.22	54.83
12	0.117	18723.57	1.27	0.64	12345.09	2.22	54.83
13	0.058	9147.22	1.07	0.65	12345.09	2.22	54.83
14	0.105	17026.66	1.12	0.63	12345.09	2.22	54.83
15	0.116	18674.90	1.29	0.64	12345.09	2.22	54.83

V. CONCLUSION

1. A CFD package (Ansys Fluent 18.1) was used for the numerical study of heat transfer characteristics of an elliptical shaped tube blank in cross flow heat exchanger and the results were validated with experimental results. The CFD results when compared with experimental results for different mass flow rate and were well within the error limits of 30 to 40%.
2. Temperature distribution contours showed that the percentage of error of outlet temperature of air from validation of CFD result with experiment is 2 to 3 %.
3. The enhancement ratio of elliptical shaped tube blank is 1.3 to 1.7 range compare to circular shaped tube blank in 9273.9 to 21000 Reynolds number range.
4. CFD analysis of model and velocity vectors showed that due to use of elliptical shaped tube blank at 45° angle of attack is delayed the boundary layer separations and it also reduce wake region formation compared to circular shape tube.
5. Velocity vector contours showed that due to orientation of tube blanks at 45° better mixing of air flow which enhance large heat transfer rate compare to 0° orientation of tube blanks.
6. The pressure distribution contours showed that reduced pressure drop due to eddies formation. But overall effect is to enhance the heat transfer rate.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

REFERENCES

- [1] S. Toolthaisong N Kasayapanand, "Effect of attack angles on side thermal and pressure drop of the cross flow heat exchangers with staggered tube arrangement", Energy Procedia 34 (2013) 417 – 429.
- [2] Talaat A Ibrahim, AbdalGomma, "Thermal performance criteria of elliptic tube bundle in crossflow", International Journal of Thermal Science 48 (2009) 2148 – 2158.
- [3] X.P. Du a, M. Zeng, "Experimental study of the effect of air inlet angle on the air side performance for cross flow finned oval tube heat exchangers", Experimental Thermal and Fluid Science 52 (2014) 146 – 155.
- [4] Lie Sun, Liang Yang, Liang Shao, "Overall thermal performance oriented numerical comparison between elliptical and circular finned tube condenser", International Journal of Thermal Science 89 (2015) 234 - 244.
- [5] Hui Han, Ya-Ling He, Yin-Shi Li, "A numerical study on compact enhanced fin and tube heat exchanger with oval and circular tube configurations", International Journal of Heat and Mass Transfer 65 (2013) 686 – 695.
- [6] Ala Hasan, Kai Siren, "Performance investigation of plain circular and oval tube evaporatively cooled heat exchangers", Applied Thermal Engineering; Vol. 24 (2014) 777 – 790.
- [7] ArashMirabdollahLavasan, HamidrezaBayat, TaherMaarefdoost, "Experimental study of convective heat transfer from in-line cam shaped tube bank in crossflow", Applied Thermal Engineering 65 (2014) 85 – 93.
- [8] Mesbah G. Khan, Amir Fartaj, David S.K. Ting, "An experimental characterization of cross flow cooling of air via an in-line elliptical tube array", International Journal of Heat and Fluid Flow 25 (2004) 636 – 648.
- [9] Cengel, Y.A. and Ghajar A. J., Heat and Mass Transfer 5th edition, Tata McGraw Hill, 446 – 450.
- [10] ShelarDigvijay, "Experimental Investigation of Elliptical Tube Bank Cross Flow Heat Exchanger with Inline Arrangement and 45° Angle of Attack", Dept. of Mechanical Engineering, (2015-16).

VI.NOMENCLATURE

T_1	Inlet temperature of water (k)
T_2	Outlet temperature of water (k)
T_3	Inlet temperature of air
T_4	Outlet temperature of air
\dot{m}	Mass flow rate (kg/sec)
ρ	Density (kg/m ³)
L	Length of duct (m)
B	Width of duct (m)
D	Diameter of semi-circular portion of tube (m)
T	Thickness of tube (m)
D_{hi}	Inside hydraulic diameter of tube (m)
D_{ho}	Outside hydraulic diameter of tube (m)
N_r	Number of rows
N_c	Number of column
A_c	Cross sectional area of tube (m ²)
A_s	Heat transfer surface area (m ²)
C_p	Specific heat (kJ/kg K)
K	Thermal conductivity (w/m K)
K	Turbulence kinetic energy (m ² /s ²)
ϵ	Turbulence dissipation rate (m ² /s ³)
μ	Dynamic viscosity (kg/ms)
ν	Kinematic viscosity (m ² /s)
Pr	Prandtl number
Re	Reynolds number
Nu	Nusselt number
H	Convective heat transfer coefficient (w/m ² K)
U	Overall heat transfer coefficient (W/m ² K)
V	Outlet air velocity (m/s)



ISSN(Online): 2319-8753
ISSN (Print): 2347-6710

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 1, January 2019

ΔT Temperature Difference ($^{\circ}C$)
Q Heat transfer (W)

Subscripts

a Air
w Water
avg Average
in Inlet