

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

Performance Analysis of Heat Exchanger using CFD

Avinash P. Shinde¹, Mahadev L. Shinde², Sagar S. Yadav³, Manasvi R. Surve⁴, A. S. Futane,
Mainak Bhaumik⁵

U.G. Student, Dept. of Mechanical Engineering, MGM CET, Kamothe, Navi Mumbai, India^{1,2,3,4}

Asst. Professor, Department of Mechanical Engineering, MGM CET, Kamothe, Navi Mumbai, India^{5,6}

ABSTRACT: The Heat exchanger is a device which is used to transfer heat from one fluid to another through a solid medium or interface. There are various types of heat exchangers available. In this paper, shell and tube type heat exchanger is selected. Our objective was to change the cross-section of the tube to improve the efficiency of the heat exchanger. Square, square with fillet and a hexagonal cross-section of tubes are selected for the study. Design of new shell and tube heat exchanger is done using standard design procedure and 3D modeling is done in Solidworks 2018. Finite Element Analysis software ANSYS Workbench 18.0 is used to perform CFD analysis under a standard working condition to find performance parameters. We found that the hexagonal cross-section provides more effective heat exchange due to an increase in the convective surface.

KEYWORDS: Heat Exchanger, Shell and Tube Type, CFD, Tube cross-section, Design, and Analysis

I. INTRODUCTION

Due to the high consumption and the reducing availability of fossil fuel resources, high thermal performance heat exchangers are subject to great interest over decades. Typically, two fluids with different temperatures circulate through the heat exchanger in natural or forced convection manners and the thermal energy is exchanged via surfaces during the process. The performance of a heat exchanger is dependent on various factors such as mass flow rate of the fluids, length of fluid travel, number and arrangement of baffle plates etc. To improve the efficiency of the heat exchanger, maximizing the surface area of the wall between two fluids and minimizing resistance to flow passing across the exchanger are two most concerned focuses. The increase of area is the most direct way to exchange more thermal energy. Performance enhancement of heat exchangers has been a hot topic for researchers all around to obtain the optimal output from a heat exchanger for the same amount of work done on it in order to conserve energy and money.

II. LITERATURE REVIEW

In this paper, a study is conducted on CFD analysis of a heat exchanger with a helically coiled tube. Due to a comparatively more surface area than a standard heat exchanger, it is proved to be more effective and efficient. [1]

In this paper, a heat transfer characteristic of different cross-sections has been conducted. They have also incorporated waviness on a channel by varying the Reynolds number and the amplitude of waviness. It was found that this different cross-section with a wavy channel provides a high heat transfer coefficient (HTC) and would be effective in thermal applications. [2]

In this paper, an attempt has been made to investigate the complex flow and temperature pattern in such a short shell and tube type heat exchanger, with and without baffles in the shell side. Heat exchangers are analyzed using CFD code OpenFOAM-2.2.0 for different mass flow rates. The effect of the flow field on the shell side heat transfer coefficient and a comparison with analytical methods are presented. [3]

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

This literature review focuses on the applications of Computational Fluid Dynamics (CFD) in the field of heat exchangers. It has been found that CFD has been employed for the following areas of study in various types of heat exchangers: fluid flow maldistribution, fouling, pressure drop and thermal analysis in the design and optimization phase.[4]

The article presents passive heat transfer enhancement method in the form of baffles to increase the energy efficiency of the heat exchanger. Most of the work focuses on the impact of geometrical parameters of the coil itself. This article successfully proves that it is possible to increase the efficiency of heat exchange in the heat exchanger shell type coil with baffle inserts. [6]

In this paper Finite Volume Method is used for numerical analysis of an internally finned axi-symmetric tube heat exchanger. In this paper the computational fluid dynamics is used for parametric study. The results obtained from the study for a steady and laminar flow of fluid under mixed flow convection heat transfer condition shows that there exists an optimum number for fins to keep the pipe wall temperature at a minimum. [7]

The paper represents that the heat transfer enhancement in a heat exchanger tube by installing fins on the outer surface of hot water tube. Design process for heat exchanger is carried out in AUTODESK INVENTOR and fluid domain is formed in ANSYS workbench. After finding the solution the results are compared between the two designs for counter flow. According to results, it concluded that in case of fin is used, effectiveness also increases due to use of fins. [8]

In this study the analytical design of the heat exchanger has been validated based on the results obtained from the Computational fluid dynamics analysis. In this paper the standard k-ε modeling is used in CFD analysis. In this paper the theoretically and analytically heat transfer rate of double pipe heat exchanger is calculated. By comparing these results, the results show that the design and analysis of the double pipe heat exchanger has been a great success. [9]

The paper represents that Experimental Study on Heat Transfer and Friction Factor in Laminar Forced Convection over Flat Tube in Channel Flow. The experiments were conducted at a flat tube in the flow direction, the five air velocity between 0.2 and 1.0 m/s, and Reynolds number based on the hydraulic diameter (Re_{Dh}) was considered from 124.5 to 622.5. The uniform heat flux supplies at the surface of the tube are 354.9, 1016.3 and 1935.8 W/m² respectively. The results show that the average Nusselt number increase with increase Reynolds number with any heat flux supply tested and also the pressure drop increased and friction factor decrease with increasing of free stream velocity. [10]

III. OBJECTIVE

1. To improve the performance of heat exchanger by changing tube geometry.
2. To test the performance using Computational Fluid Dynamics (CFD) using appropriate simulation tools.

IV. PROBLEM DEFINITION

Heat exchangers are devices which do not require power to operate themselves, but they need power in order to pump the working fluid in and out of the device's system. The Performance will be enhanced if the power required to pump the fluids is within acceptable limits according to the heat transfer occurring between the fluids, i.e. the effectiveness and efficiency of the heat exchanger. There are three goals that are normally considered in the optimal design of heat exchangers: (1) Minimizing the pressure drop (pumping power), (2) Maximizing the thermal performance and (3) Minimizing the entropy generation (thermodynamic), which are to be achieved in any possible way known or not known. Thus, an optimum method is to be evaluated, for which the performance of heat exchanger is boosted for same pumping power and ideally without any pressure drops.

V. METHODOLOGY

1. Studying the existing heat exchanger working characteristics, effectiveness, efficiency, losses etc.
2. Generating 3D model of existing heat exchanger using Solidworks software.
3. Theoretical calculations for new models.
4. Selecting of parameters for 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. 7, Issue 3, March 2018

5. Obtaining its CFD model and simulating its working condition.
6. Implementing methods that are ought to improve the performance of heat exchanger.
7. Performing CFD analysis in ANSYS Fluent on new models
8. Comparing the results with the original model.

VI. THEORETICAL CALCULATION

T_{h1} – Inlet temperature of the hot fluid
 T_{h2} – Outlet temperature of the hot fluid
 T_{c1} – Inlet temperature of the Cold fluid
 T_{c2} (CFD) – Simulation value of Inlet cold fluid
 T_{c2} – Outlet temperature of the cold fluid
 Q_h –heat loss by hot fluid
 Q_c – heat gain by cold fluid
 M_c –mass flow rate of the cold fluid (kg/s)
 C_{pc} – Specific heat capacity of cold fluid (kJ/kg-k)
 M_h – mass flow rate of the hot fluid (kg/s)
 M_{ch} – Specific heat capacity of hot fluid (kJ/kg-k)

Case1: Circular tube (original) model [12]

$$\begin{aligned} T_{h1} &= 353.15 & T_{h2} &= 345.035 \\ T_{c1} &= 293.15 & T_{c2} &=? \end{aligned}$$

$$\begin{aligned} Q_h &= Q_c \\ M_h * C_{ph} * (T_{h1}-T_{h2}) &= M_c * C_{pc} * (T_{c2}-T_{c1}) \\ 0.694*40182 (353.15-345.035) &= 1.7767*4.182 (T_{c2}-293.15) \\ T_{c2} &= 296.3198148 \text{ K} \end{aligned}$$

Case2: Square tube model

$$\begin{aligned} T_{h1} &= 353.15 & T_{h2} &= 345.508 \\ T_{c1} &= 293.15 & T_{c2} &=? \end{aligned}$$

$$\begin{aligned} Q_h &= Q_c \\ M_h * C_{ph} * (T_{h1}-T_{h2}) &= M_c * C_{pc} * (T_{c2}-T_{c1}) \\ 0.694*40182 (353.15-345.508) &= 1.7767*4.182 (T_{c2}-293.15) \\ T_{c2} &= 296.1350554 \text{ K} \end{aligned}$$

Case3: Square tube model

$$\begin{aligned} T_{h1} &= 353.15 & T_{h2} &= 345.062 \\ T_{c1} &= 293.15 & T_{c2} &=? \end{aligned}$$

$$\begin{aligned} Q_h &= Q_c \\ M_h * C_{ph} * (T_{h1}-T_{h2}) &= M_c * C_{pc} * (T_{c2}-T_{c1}) \\ 0.694*40182 (353.15-345.062) &= 1.7767*4.182 (T_{c2}-293.15) \\ T_{c2} &= 296.3092683 \text{ k} \end{aligned}$$

Case4: Square with fillet tube model

$$\begin{aligned} T_{h1} &= 353.15 & T_{h2} &= 302.784 \\ T_{c1} &= 293.15 & T_{c2} &=? \end{aligned}$$

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

$$Q_h = Q_c$$

$$M_h * C_{ph} * (T_{h1} - T_{h2}) = M_c * C_{pc} * (T_{c2} - T_{c1})$$

$$0.694 * 40182 (353.15 - 302.784) = 1.7767 * 4.182 (T_{c2} - 293.15)$$

$$T_{c2} = 312.8235543 \text{ K}$$

VII. 3D MODELING

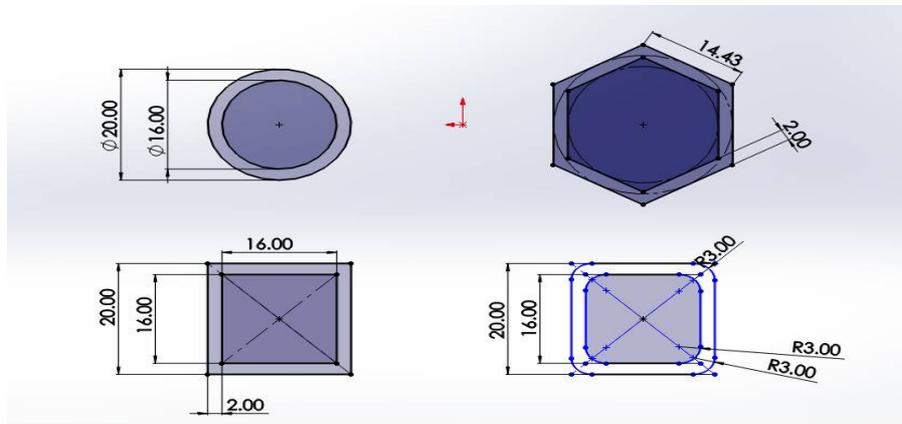


Fig 1: heat exchanger tube cross sections

Above fig shows different cross sections (circular, hexagonal, square, and square with fillet) of tube selected for study.

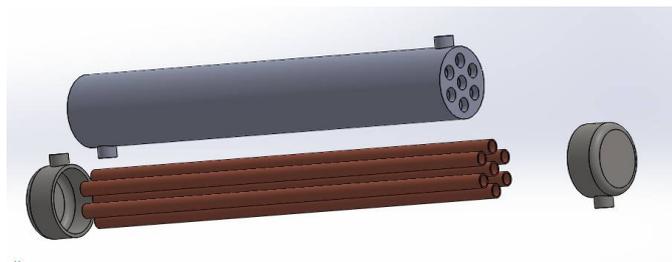


Fig 2: Circular c/s tube Heat Exchanger model

Above fig shows Heat Exchanger 3D model having circular cross section of tube.

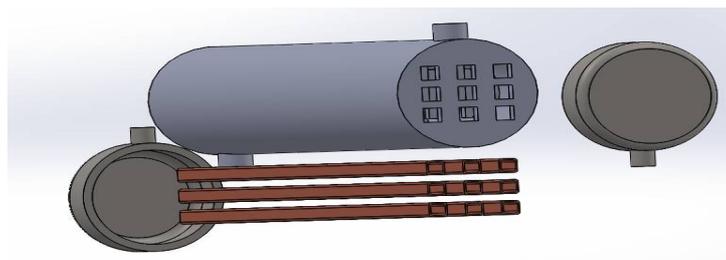


Fig 3: Square c/s tube Heat Exchanger model

Above fig shows Heat Exchanger 3D model having square cross section of tube.

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018



Fig 4: Square with fillet c/s tube Heat Exchanger model

Above fig shows Heat Exchanger 3D model having square with fillet cross section of tube.

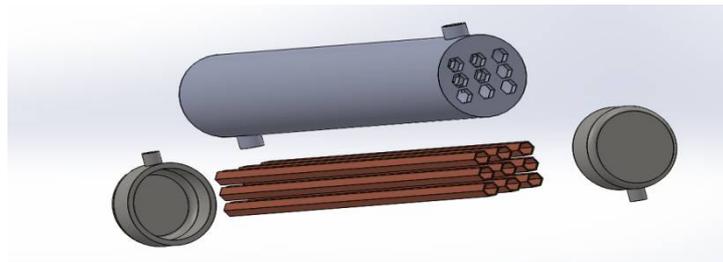


Fig 5: Hexagonal c/s tube Heat Exchanger model

Above fig shows Heat Exchanger 3D model having hexagonal cross section of tube.

VIII. CFD SIMULATION

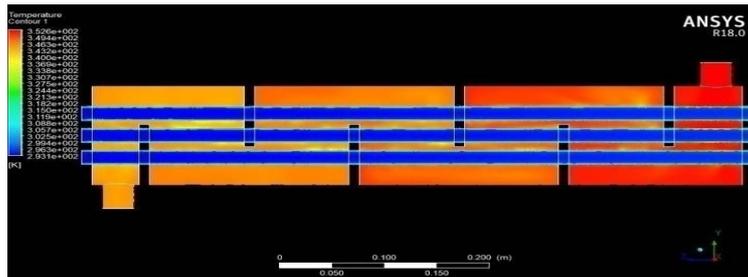


Fig 6: Temperature plot of all domains of circular c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of all domains of circular cross section tube Heat Exchanger.

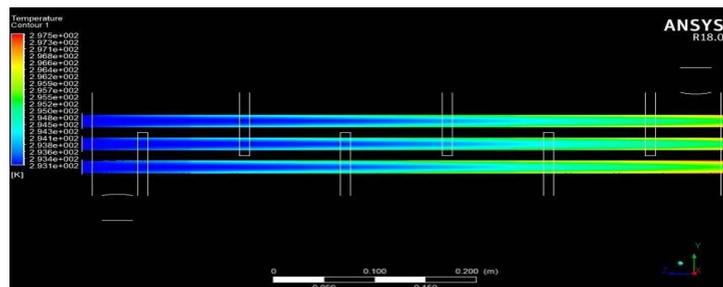


Fig 7: Temperature plot of Cold fluid of circular c/s tube H.E

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

Above fig shows CFD analysis and result of Temperature plot of Cold fluid of circular cross section tube Heat Exchanger.

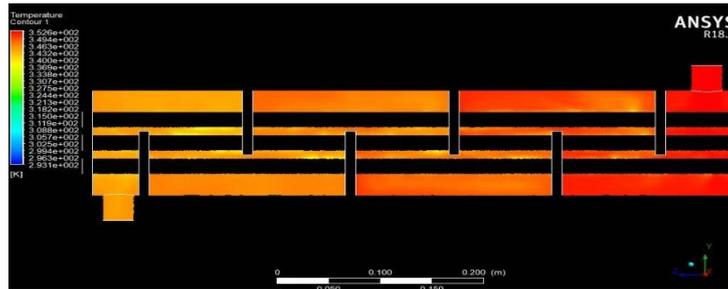


Fig 8: Temperature plot of hot fluid of circular c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of hot fluid of circular cross section tube Heat Exchanger.

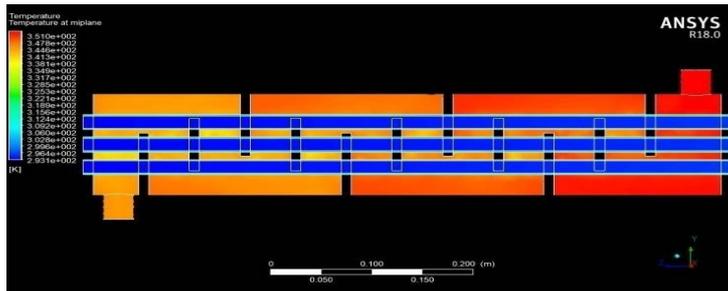


Fig 9: Temperature plot of all domains of Square c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of all domains of Square cross section tube Heat Exchanger.

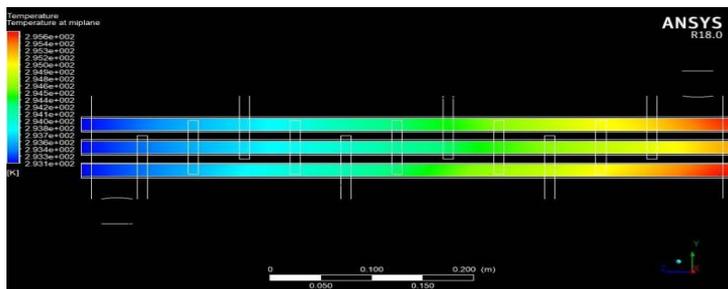


Fig 10: Temperature plot of Cold fluid of Square c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of Cold fluid of Square cross section tube 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. 7, Issue 3, March 2018

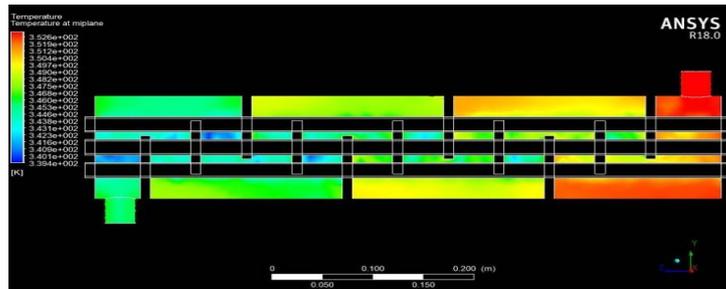


Fig 11: Temperature plot of hot fluid of Square c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of hot fluid of Square cross section tube Heat Exchanger.

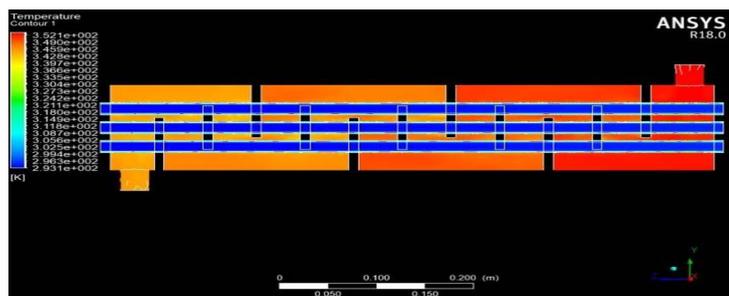


Fig 12: Temperature plot of all domains of Square with Fillet c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of all domains of Square with Fillet cross section tube H. E.

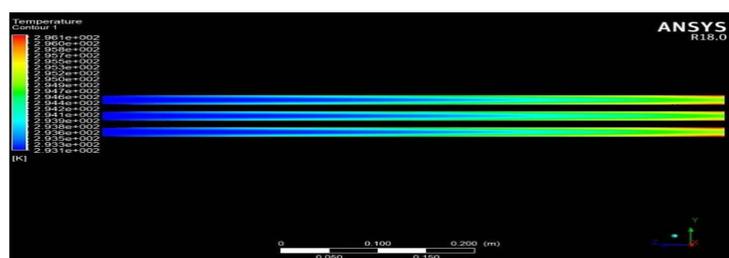


Fig 13: Temperature plot of Cold fluid of Square with Fillet c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of Cold fluid of Square with Fillet cross section tube Heat Exchanger.

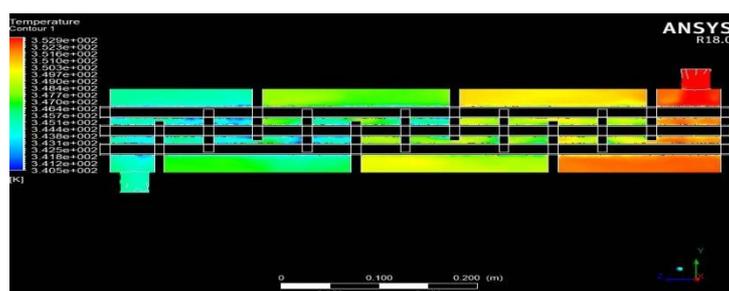


Fig 14: Temperature plot of hot fluid of Square with Fillet c/s tube H.E

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

Above fig shows CFD analysis and result of Temperature plot of hot fluid of Square with Fillet cross section tube Heat Exchanger.

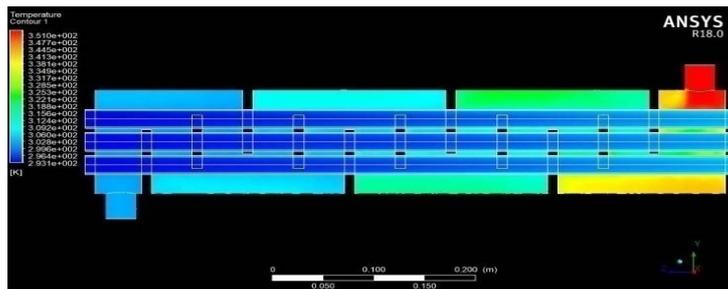


Fig 15: Temperature plot of all domains of Hexagonal c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of all domains of Hexagonal cross section tube Heat Exchanger.

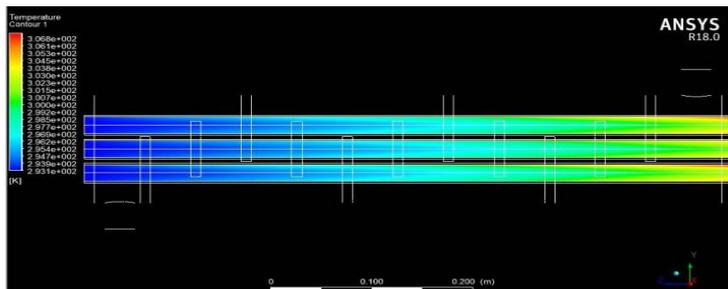


Fig 16: Temperature plot of Cold fluid of Hexagonal c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of Cold fluid of Hexagonal cross section tube Heat Exchanger.

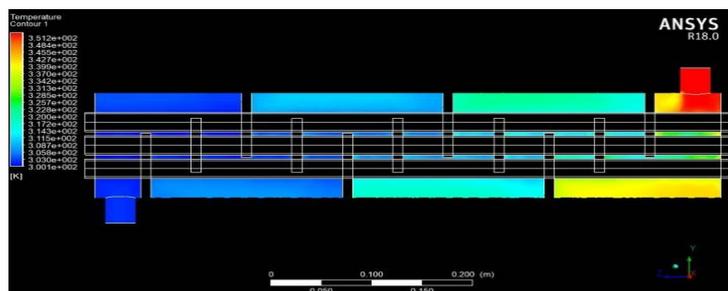


Fig 17: Temperature plot of hot fluid of Hexagonal c/s tube H.E

Above fig shows CFD analysis and result of Temperature plot of hot fluid of Hexagonal cross section tube Heat Exchanger.

IX. RESULT

Formulae

1. Percentage of difference between T-cfd and T-c2

$$[(T_{c2} - T_{c2-CFD})/T_{c2}] * 100$$

2. Heat transfer

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

$$M_h * C_{ph} * (T_{h1}-T_{h2})$$

3. Effectiveness

$$[M_h * C_{ph} * (T_{h1}-T_{h2})] / [M_h * C_{ph} * (T_{h1}-T_{c1})]$$

| Sr. No | Tube | Hot Fluid | | Cold Fluid | | | % Difference | Heat Transfer | Effectiveness |
|--------|--------------------|-----------|---------|------------|-----------|---------|--------------|---------------|---------------|
| | | T-h1 | T-h2 | T-c1 | T-c2(CFD) | T-c2 | | | |
| 1 | Circular | 353.15 | 345.035 | 293.15 | 295.372 | 296.319 | 0.31986 | 23.5522 | 0.1352 |
| 2 | Square | 353.15 | 345.508 | 293.15 | 295.635 | 22.1794 | 0.16886 | 22.1794 | 0.1273 |
| 3 | Square with fillet | 353.15 | 345.062 | 293.15 | 295.566 | 23.4738 | 0.25421 | 23.4738 | 0.1348 |
| 4 | hexagonal | 353.15 | 302.784 | 293.15 | 302.566 | 312.896 | 3.27902 | 146.177 | 0.8394 |

Table 1- Results analysis and calculation

Above table shows inlet, outlet conditions of hot and cold water in heat exchanger for all tube cross sections. % difference column shows difference between $T_{c2 (CFD)}$ and T_{c2}

Where,

T_{h1} – Inlet temperature of the hot fluid

T_{h2} – Outlet temperature of the hot fluid

T_{c1} – Inlet temperature of the Cold fluid

$T_{c2 (CFD)}$ – Simulation value of Inlet cold fluid

T_{c2} – Outlet temperature of the cold fluid

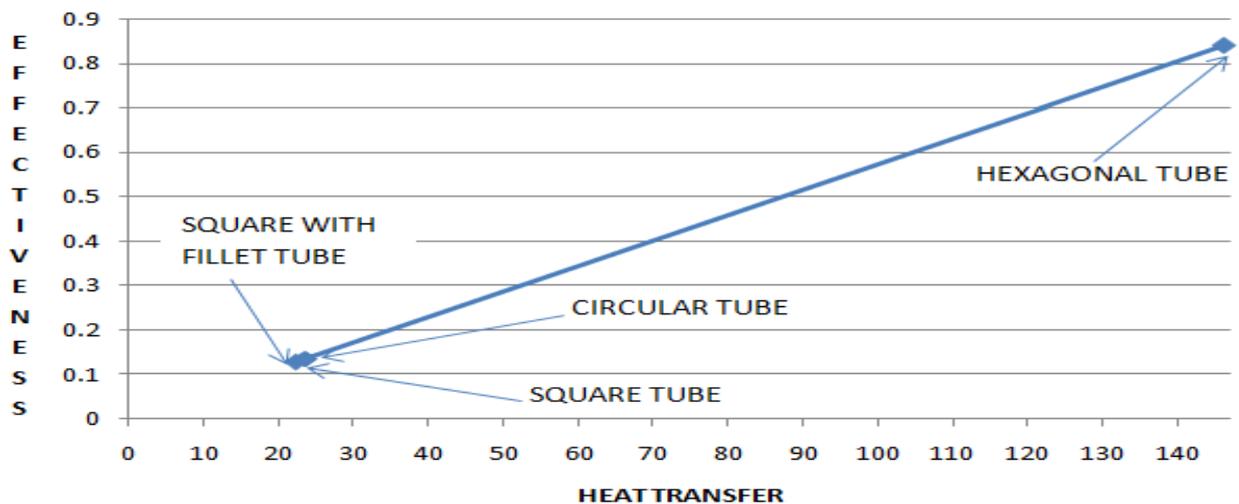


Fig 18: graph of Effectiveness VS heat transfer

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

Above fig shows graph of Effectiveness VS heat transfer which shows comparison between circular tube, square tube, square with fillet tube and hexagonal tube. From graph it is seen that heat transfer and effectiveness of hexagonal cross section tube is higher than other tube cross sections.

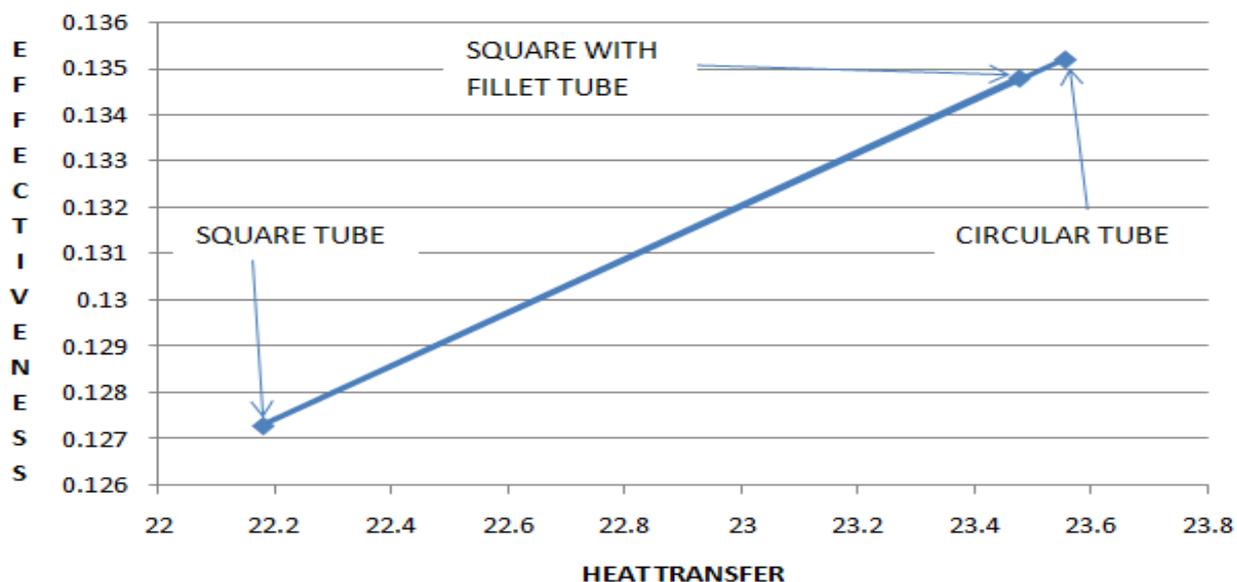


Fig 19: graph of Effectiveness VS heat transfer

Above fig shows graph of Effectiveness VS heat transfer which shows comparison between circular tube, square tube, square with fillet tube and hexagonal tube. From graph it is seen that heat transfer and effectiveness of circular cross section tube is higher than Square and Square with fillet tube cross sections. Square tube has lowest heat transfer and effectiveness.

X. CONCLUSION

From the results and data obtained from CFD we have concluded that K-w turbulence model provide better suitability to our simulation. Model with hexagonal tube provides better effectiveness than other models. Heat transfer rate in the heat exchanger with hexagonal tube also provides good heat transfer rate. Increasing effectiveness of heat exchanger increases its performance in its respective application.

ACKNOWLEDGMENT

It is our privilege to express our sincerest regards to our project Guide Prof. Mr. Mainak Bhaumik for their valuable inputs, able guidance, encouragement, wholehearted cooperation throughout the duration of our project. We thank Mr. Ratnesh Gupta for assistance with CFD and comments that greatly improved the manuscript. We pay our respects and love to our parents and for their love and encouragement throughout our career. Last but not the least we express our thanks to our friends for their support and co-operation.

REFERENCES

1. Vijaya Kumar Reddy, Sudheer Prem Kumar, Ravi Gugulothu, Kakarparthi Anuja and Viajaya Rao, "CFD Analysis of a Helically Coiled Tube in Tube Heat Exchanger", Materials Today: Proceedings 4, 2341–2349, 2017.
2. Karan Ghule, M.S. Soni, "Numerical Heat Transfer Analysis of Wavy Micro Channels with Different Cross Sections", Energy Procedia, vol. 109, pp. 471 – 478, (2017),.
3. Eshita Pal, Inder Kumar, Jyeshtharaj B. Joshi, N.K. Maheshwari, "CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles", Chemical Engineering Science, vol. 143, pp. 314–340, 2016.

International Journal of Innovative Research in Science, Engineering and Technology

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

Visit: www.ijirset.com

Vol. 7, Issue 3, March 2018

4. M. M. Bhutt, Nasir H., M. Bashir, Kanwar N. Ahmad, Sarfaraz Khan, " Review on CFD use in the various design of heat exchangers ", Applied Thermal Engineering, 32, pp.1-12, 2012.
5. Lu Ma, Ke Wang, Minshan Liu, Dan Wang, Tong Liu, Yongqing Wang, Zunchao Liu, "Numerical study on performances of shell-side in trefoil-hole and quatrefoil-hole baffle heat exchangers", Applied Thermal Engineering, vol. 123, pp. 1444-1455, 2017.
6. R Andrzejczyk, T Muszynski, "Geometrical and Thermodynamic characteristics of mixed convection heat transfer in the shell and coil tube heat exchanger with a number of baffles", Applied Thermal Engineering, volume 121, pp115-125, 2017.
7. S.K.Routa, D. N. Thatoia, A.K. Acharya, D. P. Mishra, "CFD supported performance estimation of an internally finned tube heat exchanger under mixed convection flow", Procedia Engineering 38, pp. 585-597, 2012.
8. N T Anoop.K.S, Deepak.C.S, E..P.Kuriakose, Habeeb Rahman.K.K, Karthik.K.V, "Computational fluid dynamics of the tube in tube heat exchanger with fins", International Research Journal of Engineering and Technology, volume 3, issue 4, 2016
9. J Johnson, Abdul, A Shani, Harif, H Hameed, Nithin, "Computational fluid dynamics of double pipe heat exchanger", International Journal of Science, Engineering and technology research, volume 4, issue 5, 2015
10. Tahseen Ahmad Tahseena, M.M. Rahman and M. Ishaka, "Experimental study on Heat transfer and friction factor in laminar forced convection over flat tube over channel flow", International conference on thermal engineering, volume 105, pp. 46-55, 2015
11. Shrikant, R. Sivakumar, N. Anantharaman, M. Vivekenandan, "CFD simulation study of shell and tube heat exchangers with different baffle segment configurations", Applied Thermal Engineering, Volume 108, Pages 999-1007, 2016.
12. Mustapha Mellal, Redouane Benzeguir, Djamel Sahel, Houari Ameer, "Hydro-thermal shell-side performance evaluation of a shell and tube exchanger under different baffle arrangement and orientation", International Journal of Thermal Sciences, volume 121, page no 138-149, 2017.