OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Experimental Investigation on Heat Transfer Analysis in a Cross flow Heat Exchanger with Waved Baffle Plates by using FLOEFD

V. Chandrikha[1], D. Santha Rao[2], P. Sriivasa Rao[3]
[1] (M.Tech Student, Mech. Dept. BVC Engineering College, Odalarevu)
[2] (Associate Professor, Mechanical Department, BVC Engineering College, Odalarevu)
[3] (Assistant Professor, Mechanical Department, Sai Spurthi Inst. of Tech, B. Gangaram)

Abstract: *Heat exchanger is devices used to exchange the heat between two liquids that are at different temperature .These are used as a reheated in many industries and auto mobile sector and power plants. The main aim of our project is thermal analysis of heat exchanger with waved baffles for different types of materials at different mass flow rates and different tube diameters using FLOEFD software and comparing the results that are obtained. The work is a simplified model for the study of thermal analysis of shell-and-tubes heat exchangers having water as cold and hot fluid. Shell and Tube heat exchangers are having special importance in boilers, oil coolers, condensers, pre-heaters. They are also widely used in process applications as well as the refrigeration and air conditioning industry. The robustness and medium weighted shape of Shell and Tube heat exchangers make them well suited for high pressure operations. The project shows the best material, best boundary conditions and parameters of materials we have to use for better heat conduction. For this we are chosen a practical problem of counter flow shell and tube heat exchanger having water, by using the data that come from cfd analysis. A design of sample model of shell and tube heat exchanger with waved baffles is using Pro-e and done the thermal analysis by using FLOEFD software by assigning different materials to tubes with different diameters having different mass flow rates and comparing the result that obtained from FLOEFD software.*
Keywords: *Heat Exchanger, Creo Design, Materials (Al-6061, Copper and Steel), FLOEFD Analysis, Mass flow rates, dimensions of Materials, heat transfer rate.*

I. Introduction

A heat exchanger is a device that is used for transfer of thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at differing temperatures and in thermal contact, usually without external heat and work interactions. The fluids may be single compounds or mixtures. Typical applications involve heating or cooling of a fluid stream of concern, evaporation or condensation of a single or multi component fluid stream, and heat recovery or heat rejection from a system. In other applications, the objective may be to sterilize, pasteurize, fractionate, distill, concentrate, crystallize, or control process fluid. In some heat exchangers, the fluids exchanging heat are in direct contact. In other heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transient manner.

In most heat exchangers, the fluids are separated by a heat transfer surface, and really they do not mix. Such exchangers are referred to as the direct transfer type, or simply recuperate. In contrast, exchangers in which there is an intermittent heat exchange between the hot and cold fluids via thermal energy storage and rejection through the exchanger surface or matrix are referred to as the indirect transfer type or storage type, or simply regenerators. Such exchangers usually have leakage and fluid carryover from one stream to the other. Many types of heat exchangers have been developed to meet the widely varying applications. Bank of tubes are found in many industrial processes and in the nuclear industry, being the most common geometry used in heat exchanger. The heat is transferred from the fluid inside the tubes to the flow outside them.

In the shell and tube heat exchanger, the cross flow through the banks is obtained by means of baffle plates, responsible for changing the direction of the flow and for increasing the heat exchange time between fluid and the heated surfaces. Numerical analysis of the laminar flow with heat transfer between parallel plates with baffles was performed by Kelkar and Patankar [2]. Results show that the flow is characterized by strong deformations and large recirculation regions. In general, Nusselt number and friction coefficient (FR) increase with the Reynolds number. Measurement using LDA technique in the turbulent flow in a duct with several baffle plates were performed by Berner et al. [3], with the purpose of determining the number of baffles necessary for obtaining a periodic boundary condition and the dependence on Reynolds number and the geometry. Results showed that with a Reynolds number of 5.17×103, four baffles are necessary for obtaining a periodic boundary condition. By increasing the Reynolds number to 1.02×104, a periodic boundary condition is obtained with three baffles. A significant amount of research has focused both on channels with internal obstructions and tortuous channels, to determine the configurations that lead to the most vigorous mixing and highest rate of heat transfer. Popiel and Van Der Merwe [4] and Popiel and Wojkowiak [5] who studied experimental pressure drops for geometries with an undulating sinusoidal or U-bend configuration. In these papers, the effects of Reynolds number, curvature, wavelength and amplitude on the friction factor were investigated in laminar and low Reynolds number turbulent flow. An interesting observation made by these authors is that when the friction factor is plotted against the Reynolds number, there is either no definite transition from laminar to turbulent flow, or a delayed transition relative to that of a straight pipe. It is hypothesized by Popiel and Van der Merwe [4] that a smooth transition to turbulence occurs due to the secondary flows produced within the complex geometry. Dean [6] originally observed that the mixing effects of these secondary flows are steadily replaced by the development of turbulent secondary flow. A method to study fully developed flow and heat transfer in channels with periodically varying shape was first developed by Patankar et al. [7] for the analysis of an offset-plate fin heat exchanger.

Their method takes advantage of the repeating nature of the flow field to minimize the extent of the computational domain. The method of Parankar et al. [7] assumes that for a periodic geometry, the flow is periodic with a prescribed linear pressure gradient being applied to drive the flow. The outlet velocity field and its gradient are wrapped to the inlet to produce periodic boundary conditions. Flow velocities within the geometry are then calculated using momentum and mass conservation equations, assuming constant fluid properties.Webb and Ramadhyani [8] and Park et al.[9] analyzed fully developed flow and heat transfer in periodic geometries following the method of Patankar. Webb and Ramadhyani [8] studied parallel plate channels with transverse ribs; they presented a comparison with the performance of a straight channel, and reported an increase in both the heat transfer rate and pressure drop as the Reynolds number is increased. Park et al. [9] incorporated optimization of the heat transfer rate and pressure drop into their study of the flow and thermal field of plate heat exchangers with staggered pin arrays. N.R. Rosaguti, D.F. Fletcher, and B.S. Haynes [10] analyzed fully developed flow and Heat Transfer in geometries that are periodic in the flow direction. They have studied laminar flow in serpentine duct of circular cross section with a constant heat flux applied at the walls, they measured the performance of serpentine channel by comparing pressure drop and rate of heat transfer in these channels to that achieved by fully developed flow in a straight pipe equal path length. Flow characteristics within such channels are complex, leading to high rates of heat transfer, whilst low pressure loss is maintained. Dean vortices act to suppress the onset of recirculation around each bend and are the main contributing factor to these high levels of heat transfer performance, and low normalized friction factor. For L/d=4.5, Rc/d=1 and Pr=6.13 two of vortices are observed at Reynolds Number above 150. This flow structure occurs immediately after bends that turn in an opposite direction to the one previous. The influence of L/d on Heat Transfer and pressure drop has been shown for a fixed Reynolds Number. Increasing L/d increases the rate of heat transfer and decreases the pressure drop relative to that of fully developed flow in a straight pipe.L.C. Demartini, H.A. Vielmo, and S.V. Moller [11] investigated the numerical and experimental analysis of the turbulent flow of air inside a channel of rectangular section, containing two rectangular baffle plates, where the two plates were placed in opposite walls.

The scope of the problem is to identify the characteristics of the flow, pressure distribution as well as the existence and extension of possible recirculation in Heat Exchanger. The geometry of the problem is a simplification of the geometry baffle plate found in Shell- and- tube Heat Exchanger. The most important features observed are the high pressure regions formed upstream of both baffle plates and the extension of the low pressure regions on the downstream region. The latter are strongly associated with the boundary layer separation on the tip of the baffle plates, which is also influenced by the thickness of the baffle plates. Low and high pressure regions are associated to recirculation regions. The most intense is that occurring downstream of the second baffle plate, responsible for the high flow velocities observed at the outlet of the test section, creating a negative velocity profiles which introduces mass inside the test section through the outlet. Numerical studies of unsteady laminar flow heat transfer in grooved channel flows of especial relevance to electronic system was performed by Y.M. Chung & P.G. Tucker [12]. The validity of a commonly used periodic flow assumption is explored. Predictions for Re=500 show the flow typically can become periodic by around the fifth groove. Hence, when modeling IC rows on circuit boards the popular periodic flow assumption might not be valid for significant area. (13).Present work attempts to investigate the heat transfer performance of shell and tube heat exchanger designed by Creo in FLOEFD.

II. Experimental set up and procedure

2.1 Experimental set up

Shell and Tube heat exchanger is designed by Creo software, actually PTC Creo Parametric is the standard in 3D CAD, featuring state-of-the-art productivity tools that promote best practices in design while ensuring compliance with your industry and company standards. PTC Creo Parametric provides the broadest range of powerful yet flexible 3D CAD capabilities to help you address your most pressing design challenges including accommodating late stage changes, working with multi-CAD data and electromechanical design A scalable offering of integrated, parametric, 3D CAD, CAID, CAM, and CAE solutions allows you to design faster than ever, while maximizing innovation and quality to ultimately create exceptional products. The following figure shows the assembly of shell and tube heat exchanger model in Creo software.

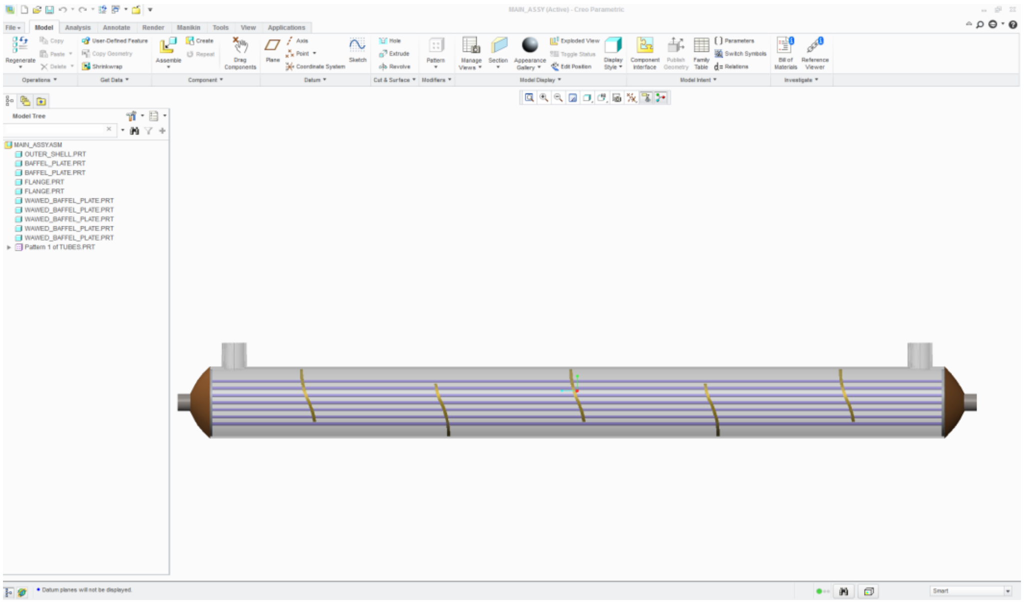

Fig. 1: assembly of shell and tube heat exchanger in creo

Heat transfer analysis predicted by using FLOEFD. Actually FLOEFD is a fluid flow and heat transfer analysis software that is fully integrated in Creo Elements/ProE and is based on the proved computational fluid dynamics (CFD) technology. Unlike other CFD software, FLOEFD works directly with native Creo Elements/Pro geometry in order to keep pace with on-going design changes. It has the same "look and feel" as Creo Elements/Pro itself, so you can focus on solving the problem instead of learning a new software environment. FLOEFD can reduce simulation time by as much as 65 to 75 percent in comparison to traditional CFD tools due to its adoption of Concurrent CFD technology and enables users to optimize product performance and reliability while reducing physical prototyping and development costs without time or material penalties.

Designed by engineers for engineers, FLOEFD is widely used in many industries and for various applications, where design optimization and performance analysis are extremely important, such as valves and regulators, hydraulic and pneumatic components, heat exchangers, automotive parts, electronics and many others.

To perform an analysis, you just need to open your model and go through the following steps:

1) Create a FLOEFD project describing the most important features and parameters of the Problem. You can use the Wizard to create the project in a simple step-by-step process.
2) Specify all necessary Input Data for the project.
3) Run the calculation. During this process, you can view the calculation progress on the Solver Monitor.
4) Analyze the obtained Results with powerful results processing tools available in FLOEFD.

The following figures show the assembly of shell and tube heat exchanger 3-D model with waved baffle plates in FLOEFD software.

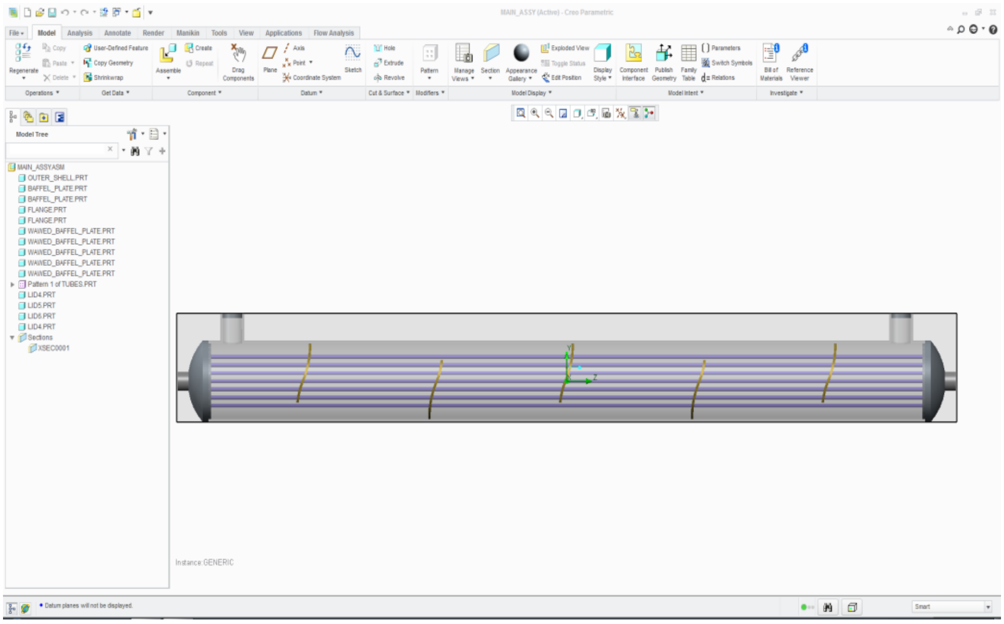

Fig.2: 3-d model of tube-shell heat exchanger with waved baffles.

The following figures show the thermal analysis of heat exchanger in FLOEFD.

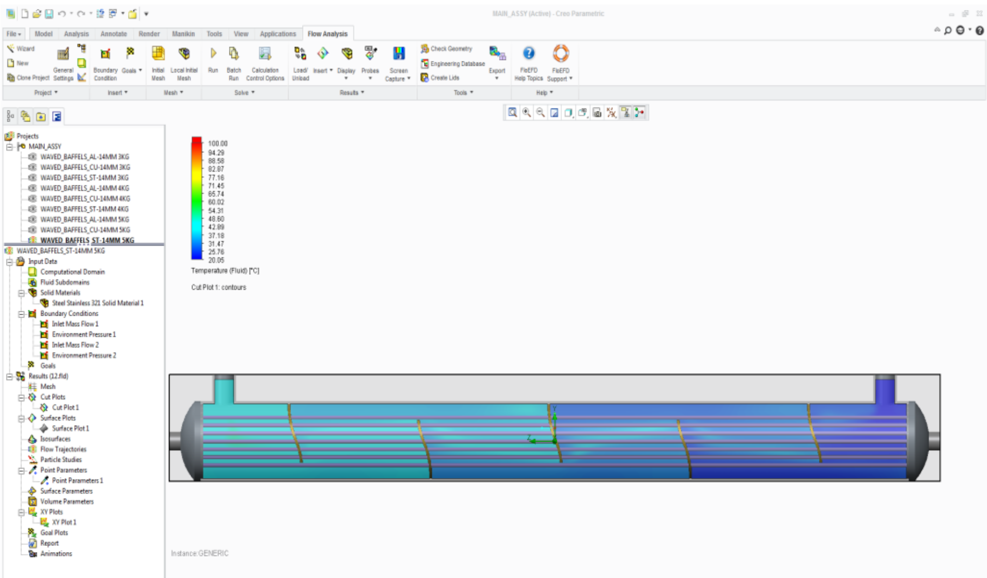

Fig. 3: thermal analysis in floefd

Table1. Specifications of the Heat Exchanger

Specifications of the Heat Exchanger		
S.I No	**Shell dimensions(mm)**	**Tube dimensions(mm)**
Length	300	300
Diameter	300	10,12,14
Thickness	1	1

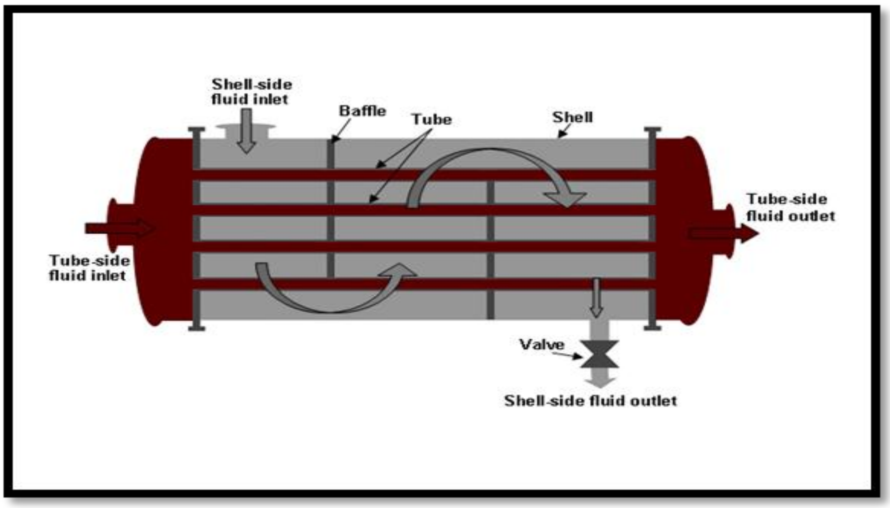

Fig.4. shell and tube heat exchanger

2.2 Experimental Procedure by using FLOEFD.

Shell and Tube heat exchanger Experimental working analysis in FLOEFD is summarized as the following steps.

1) First of all open the 3d model in floefd.
2) Open the flow analysis tab
3) Before opening wizard create lids at all open conditions. Otherwise the error note will appeared.
4) After creating lids, click on new wizard. A small window is appeared then enters the project title and set the dimensions as per requirement.
6) Click on internal analysis type. Because of the fluid is flowing inside the shell. And click on box as per requirement. Then selects the analysis medium
7) Selecting water as medium enters the wall conditions and specifies the environment conditions, set the mesh type as per requirement. Click finish.
8) After creating wizard, assign the material and boundary conditions And RUN the project.

After complete running of the project in FLOEFD, load the results in the tabular form. Finally the results are shown in the graphical presentation is as follows.

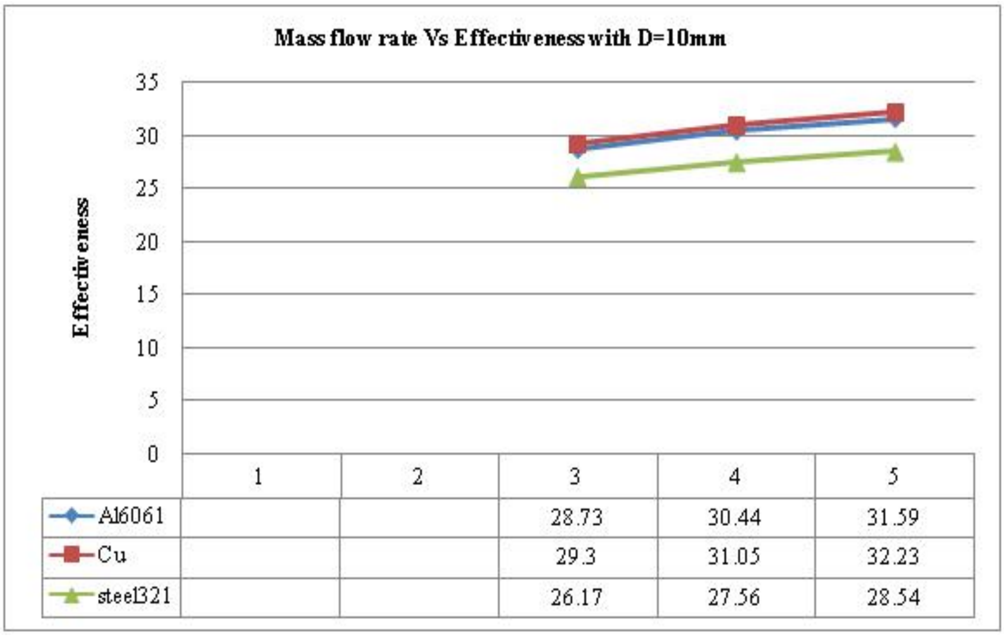

Fig 6: variation of mass flow rate with effectiveness (d=10mm)

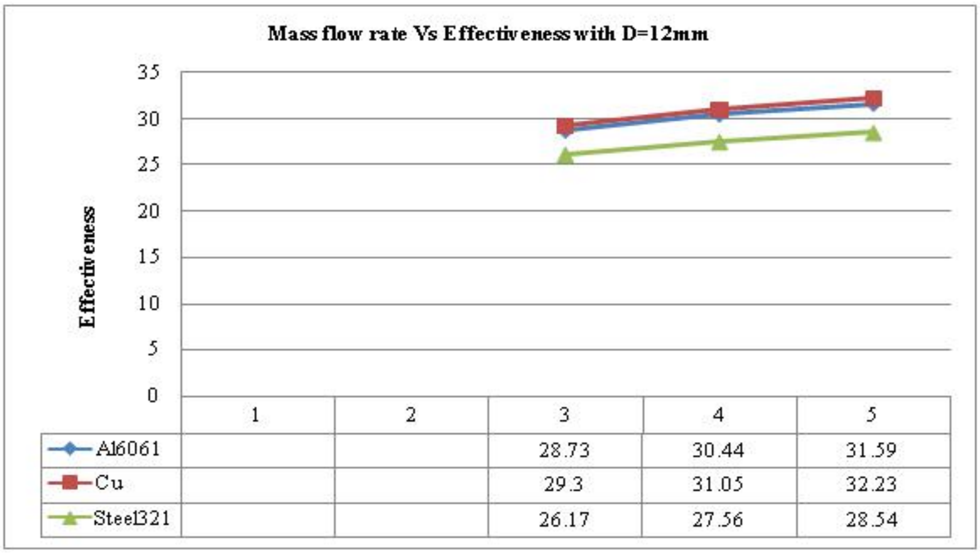

Fig.7. variation of mass flow rate with effectiveness (d=12mm)

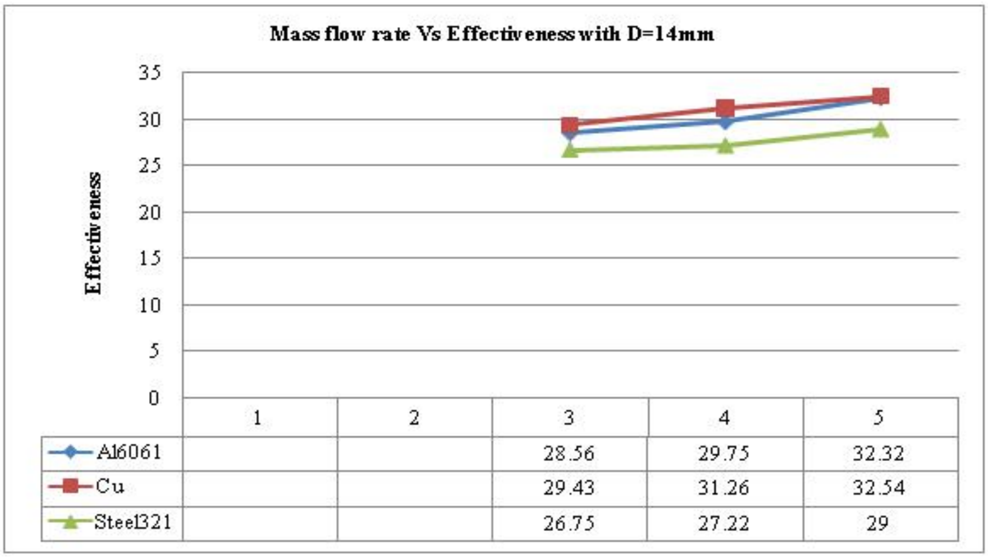

Fig. 5: variation of mass flow rate with effectiveness (d=14mm)

III. Results and Discussion

From above the graphs, we can say that the following 3 type of results these are

1. Material Discussion:

Copper has higher thermal conductivity than aluminum, steel. Comparing above results due to less density of aluminum than copper and steel the heat transfer rate is more in aluminum than steel. So it's better to use aluminum 6061, copper heat exchanger.

2. Dimensions Discussion:

Comparing the results of above three diameters i.e. 10mm 12mm 14mm, 14mm diameter tube has better heat transfer rate than remaining two. Because the 14mm diameter tube has more area in contact with fluid resulting better performances. So it's better to use large diameter tube (i.e. more area contact with fluid).

3. Boundary condition Discussion:

Comparing the above results the fluid which has different mass flow rates i.e. 3kg/s, 4kg/s, 5kg/s Have better performance at 5kg/s. Because due to medium mass flow rate the velocity of fluid is medium and more time the fluid particle is in contact with heat area of metal and absorbs heat. So it's better to maintain medium mass flow rate 5kg/s.

IV. Conclusions

The conclusions deriving from present experimental investigation to evaluate the After completing the analysis of counter flow heat exchanger of different materials, at different mass flow rates with different diameter of tubes by using FLOEFD software. We can conclude that, Aluminum 6061, copper heat exchanger with medium mass flow rate (5kg/s) and the tubes with more area in contact with hot fluid (14mm) has better heat transfer performance compared to other materials, mass flow rates and diameters.

Acknowledgement

The Authors also would like to thank the Sanya Technology Pvt. Limited, Hyderabad, A.P. and India. For supporting this research, and also thank to our guides, they are given valuable suggestions to completion of this project.

REFERENCES

[1] Patankar, S.V. and spalding. D.B. (1974), " A calculation for the transient and steady state behaviour of shell- and-tube Heat Exchanger". Heat transfer design and theory sourcebook. Afgan A.A. and Schluner E.U.. Washington. Pp. 155-176.

[2] KelKar, K.M and Patankar, S. V., 1987" Numerical prediction of flow and Heat transfer in a parallel plate channel with staggered fins", Journal of Heat Transfer, vol. 109, pp 25-30.

[3] Berner, C., Durst, F. And McEligot, D.M., "Flow around Baffles", Journal of Heat Transfer, vol. 106 pp 743-749.

[4] Popiel, C.o & Vander Merwe, D.F., "Friction factor in sine-pipe flow, Journal of fluids Engineering", 118, 1996, 341-345.

[5] Popiel, C.O & Wojkowiak, J., "friction factor in U-type undulated pipe flow, Journal of fluids Engineering", 122, 2000, 260-263.

[6] Dean, W. R., Note on the motion of fluid in a curved pipe, The London, Edinburgh and Dublin philosophical Magazine, 7 (14), 1927, 208-223.

[7] Patankar, S.V., Liu, C.H & Sparrow, E.M., "Fully developed flow and Heat Transfer in ducts having streamwise periodic variation in cross-sectional area", Journal of Heat Transfer, 99, 1977, 180-186.

[8] Webb, B.W & Ramdhyani, S., "Conjugate Heat Transfer in an channel with staggered Ribs" Int. Journal of Heat and Mass Transfer 28(9), 1985,1679-1687.

[9] Park, K., Choi, D-H & Lee, K-s., "design optimization of plate heat exchanger with staggered pins", 14th Int. Symp.on Transport phenomena, Bali, Indonesia, 6-9 July 2003.

[10] N.R. Rosaguti, D.F. Fletcher & B.S. Haynes "Laminar flow in a periodic serpentine". 15th Australasian fluid mechanics conference. The University of Sydney, Sydney, Australia 13-17 December 2004.

Books

[1] Textbook Heat and Mass transfer by RAJPUT

[2] Textbook Heat &Mass transfer by S.DOMAKUNDWAR & S.C. ARORA