Modeling and Heat Transfer Analysis of Double Pipe Heat Exchanger Using Computational Fluid Dynamics (CFD)

Murali Ponnusamy

Dr.A.P.J.Abdul kalam Centre for Advanced research, Adhi College of Engineering and Technology, Kanchipuram, Tamil Nadu, India
murali.mech@adhi.edu.in

Abstract

Present study demonstrate that a two dimensional analysis of heat transfer distinctive of double pipe heat exchanger (also called pipe-in-pipe exchanger) beneath different parameters such as mass flow rate and direction, tube materials, with fins etc., used to predict the performance. However, heat exchanger is numerically modeled for tempestuous as well as laminar fluid flow condition. Notably, Heat transfer rate, overall heat transfer co-efficient, Reynolds number, Nussle numbers are to be calculated by analytical method aimed at both parallel along with counter flow heat exchanger. On the other hand, Computational Fluid Dynamics (CFD) analysis has been made to visualize the temperature as well as velocity distribution. The computational simulation model of heat exchanger will be developed with FLUENT and validated with experimental results.

Keywords: Heat exchanger, Laminar and turbulent flow, Heat transfer characteristics, CFD, Modelling and analysis.

1. INTRODUCTION

The heat exchanger is an expedient that is used to transfer thermal energy among two or more fluids or a solid surface and a fluid, or solid particulates and fluid, at different temperatures and thermal contact. Heat exchangers raise or lower the temperature of these streams by transferring heat to from the stream. Transfer of heat happens by three principle means: Radiation, Conduction and Convection. In the use of heat exchangers radiation does take place. However, the comparison of conduction and convection to radiation does not play a major role. Conduction occurs as the heat from the higher temperature, fluid passes through the solid wall to maximize the heat transfer, and the wall should be thin and made of a very conductive material. The biggest contribution to heat transfer in a heat exchanger is made through convection. In a heat exchanger forced convection allows for the transfer of heat of one moving stream to another moving stream. It maintains a temperature gradient between the two fluids. Heat exchangers are used in wide variety of applications; it includes power production process, chemical and food industries, electronics, environmental engineering, waste heat recovery and manufacturing industry.

Heat exchanger may be classified according to following main criteria: 1) Recuperates and regenerators 2) Transfer process: Direct contact and Indirect contact 3) Geometry of construction: tubes, plates, extended surfaces 4) Heat transfer mechanism: single phase and two phase 5) Flow arrangements: Parallel, counter and cross flow.

CLASSIFICATION ACCORDING TO TRANSFER PROCESSES

Indirect-Contact Heat Exchangers: In an indirect-contact heat exchanger, the fluid streams remain separate and the heat transfers continuously through an impervious dividing wall or into and out of a wall in a transient manner. Thus ideally there is no direct contact between thermally interacting fluids. **Direct-Contact Heat Exchangers:** In a direct-contact exchanger, two fluid streams come into direct contact, exchange heat, and are then separated. A common application of a direct-contact exchanger involves mass transfer in addition to heat transfer, such as in evaporative cooling and rectification; applications involving only balanced heat transfer are rare. The enthalpy of phase change in such an exchanger generally represents a significant portion of the total energy transfer. The phase change generally enhances the heat transfer rate. Compared to indirect contact recuperators and regenerators, in direct-contact heat exchangers, (1) very high heat transfer rates are achievable, (2) the exchanger construction is relatively inexpensive, and (3) the fouling problem is generally non-existent, due to the absence of a heat transfer surface (wall) between the two fluids. However, the applications are limited to those cases where a direct contact of two fluid streams is permissible. These exchangers may be further classified as follows.

a. Immiscible Fluid Exchanger
b. Gas-liquid exchangers
c. Liquid–Vapor Exchangers

II. LITERATURE REVIEW

Gabriela Huminic et al., presented work a three-dimensional analysis is used to study the heat transfer topographies of double-tube helical heat exchangers using nano fluids under laminar flow conditions. CuO and TiO2 nanoparticles with diameters of 24 nm spread or distributed in water with volume concentrations of 0.5–3 vol. % are used as a working fluid. The mass flow rate of the nano fluid from the inner tube was kept and the mass flow rate of the water from the annulus was set at either half, full or double the value. The variations of the nano fluids and water temperatures, heat transfer rates and heat transfer coefficients along inner and outer tubes are shown in the paper. Further, effects of nanoparticles concentration and of the Dean number on the heat transfer rates and heat transfer coefficients are presented. The results show that for 2% CuO nanoparticles in water and same mass flow rate in inner tube and annulus, the heat transfer rate of the nano fluid was approximately 14% greater than of pure water and the heat transfer rate of water from annulus than through the inner tube flowing nano fluids was approximately 19% greater than for the case which through the inner and outer tubes flow water.

Timothy J. Rennie et al., experimented double-pipe helical heat exchanger was numerically modeled for laminar fluid flow and heat transfer characteristics under different fluid flow rates and tube sizes. Two different tube diameters were used. The overall heat transfer coefficients were calculated for both parallel flow and counter flow. Validation of the simulations was conducted by comparing the Nusselt numbers in the inner tube with those found in literature; the results fell within the range found in the literature. The greatest thermal resistance was found in the annular region. The annulus Nusselt number was correlated with a modified Dean number, and showed a strong linear relationship.

Wen-Lih Chen et al., this paper, a numerical study on the flows in parallel and counter flow double tube heat exchangers with the inner tubes being either alternating horizontal or vertical oval cross section pipes or circular pipes is presented. The results include temperature and pressure contours and velocity vectors at several selected cross sections, axial averaged Nusselt number distributions and distributions of overall heat transfer coefficient and heat transfer enhancement factor versus three different parameters. The computation shows that the introduction of the inner alternating oval tube produces axial vortices in both the inner and outer tube flows, and the tube's heat transfer performance is improved as a result. In general, the counter flow arrangement returns a higher level of overall heat transfer coefficient than the parallel flow arrangement.

Bergles et al., heat transfer and pressure drop data for straight and spiral finned tubes of fin heights from 0.77 to 3.3mm with water as the working fluid was investigated. The Reynolds number based on the hydraulic diameter ranged from around 1, 500 to 50, 000. They found an earlier transition from laminar to turbulent flow and their friction factor data indicated that the smooth tube friction factor correlations could also be used for the tested finned tubes in the turbulent region. The heat transfer coefficients were found to be up to twice that of comparable smooth tubes. From their heat transfer data, they concluded that the hydraulic diameter approach is effective for correlation only in the case of straight fins of moderate heights.

In this paper, heat exchanger (HE) is considered and computational fluid dynamics (CFD) is used to analyze its evaporator's performance and based on it, will be try to increase the thermal efficiency and to optimize the distribution of fluid flow in this type of heat exchangers. The CFD principles which are made use in this article are effective and appropriate methods that using finite volumes to solve the processes that consist of transport phenomena. The necessary numerical computations are accomplished by Fluent (the CFD solver program) and the results are given in graphical representation. This analysis is done for an existing HE with the specified conditions of inlet flow to evaporator and finally the comparison and conclusion are presented.

III. DESIGN OF DOUBLE PIPE HEAT EXCHANGER

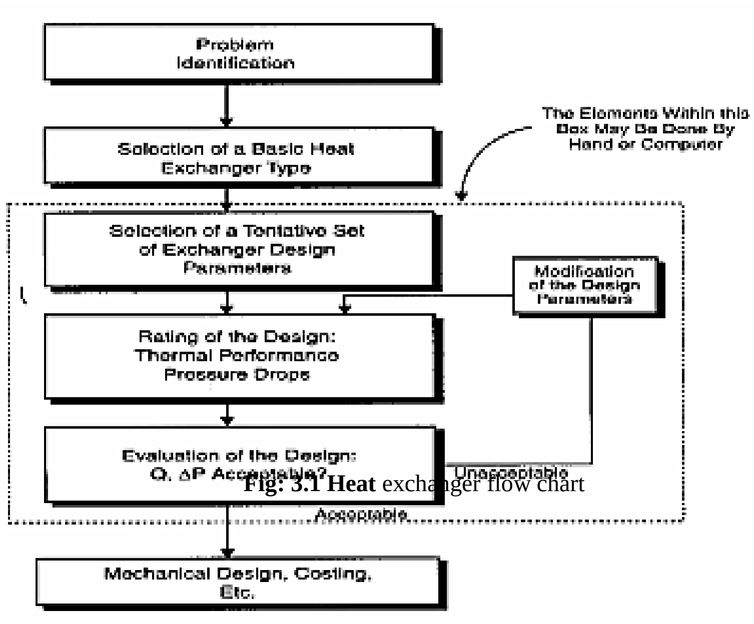


Fig: 3.1 Heat exchanger flow chart

BASIC DESIGN PROCEDURE

Heat exchanger must satisfy the heat transfer requirements (design or process needs) and allowable pressure drop (pumping capacity and cost). The steps in designing a heat exchanger **c**an be listed as: 1) Identify the problem 2) Select a heat exchanger type 3) Calculate/Select initial design parameters 4) Calculate thermal performance and pressure drops 5) Evaluate the design 6) Modification of design parameters. The step in typical design parameters are 1) Define the heat duty (Q) ex, heat to be transfer 2) Based on fluid flow rate, temperature specific heats 3) Select the overall heat transfer coefficient (U) 3) Calculate the mean temperature difference (LMTD) 4) Calculate the area required from equation (Ao).

a. Calculation of (Heat duty)

Heat duty can be calculated for both fluids. Shell and tube side mass flow rates is known, specific heat at given fluids is known .unknown temperature values is find out from the second formula , finally find out the heat duty (Q)

$$Q = mC_p\,(T_i - T_o)\text{ hot fluid} = mC_p\,(T_i - T_o)\text{ cold fluid}$$

b. Shell side heat transfer coefficient (h_a)

The heat transfer coefficient outside the tube is referred to as the shell-side heat transfer coefficient. Then, the heat transfer coefficient can be based on the equivalent diameter.

$$h_a = \frac{Nu_a . k}{\mathrm{D_e}}$$

h_a– outside heat transfer coefficient

Nu—nusselt number

k – Thermal conductivity

De -equivalent diameter

c. Tube side heat transfer coefficient (h_i)

Theoretical calculations for the case of fully developed turbulent flow with constant properties in a circular tube with constant heat flux boundary conditions. Tube side heat transfer based on the materials and velocity of fluids

$$h_i = \frac{Nu_i . k}{\mathrm{d_i}}$$

h_i –inside heat transfer coefficient
Nu—nusselt number
k – Thermal conductivity

d. Estimation of overall heat transfer coefficient (U)

The overall heat transfer coefficient Uo based on the outside diameter of tubes can be estimated from The individual heat transfer coefficients (h) Shell wall, outside & inside tube fouling resistances (Rw, Rfo, Rfi).

$$U = \frac{1}{\frac{d_o}{d_i h_i} + \frac{d_o \ln(d_o/d_i)}{2k} + \frac{1}{h_o}}$$

h_i – Inside heat transfer co-efficient
h_o - Outside heat transfer co-efficient

e. Mean temperature difference (LMTD)

The temperature difference between hot and cold fluids in the heat exchanger varies from point to point. In addition various mode of heat transfer are involved. Therefore based on concept of appropriate mean temperature difference, also called mean temperature difference

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln(\frac{\Delta T_1}{\Delta T_2})}$$

f. Calculate the surface area from equation

Surface area calculate from the below equation and all values are known, from that we find out the surface area of shell and tube heat exchanger. It doing important role in heat transfer.

$$A_s = 2\pi d_o L$$

Table 3.1 Dimensions of heat exchanger

Tube Diameter(m)	D_i	d_i	d_o
Copper	0.040	0.012	0.015
Aluminum	0.045	0.012	0.015

IV. MODELING AND ANALYSIS

Geometries for the heat exchanger were created in Gambit and exported as mesh files. Two different models were created and the values are given. Each of the heat exchangers had a length 1.8m and the geometry files were imported into a commercial computational fluid dynamics. Inlets and outlets were located at each end of the tube. The boundary conditions associated with the inlets specified the inlet velocities in the axial direction. The outer surface of the heat exchanger was set to be adiabatic and the inner tube was set to allow conductive heat flow through the tube. Fluid properties and material properties of the heat exchanger are given in Table 2.Simulations were performed using four different mass flows in the inner tube (0.0285, 0.0354, 0.0379, and 0.0397 kg /s). These resulted in heat transfer rate For each inner flow rate, also were performed using four different mass flows in shell side (0.0299, 0.0367, 0.0396, 0.0415) these resulted in heat transfer rate outside mass flow rate.

Three trials were performed with annulus mass flow rates that were 1/2, 1, and 2 times the inner flow rate. For example, the three annulus mass flow rates associated with the inner mass flow rate of (0.0285, 0.0354, 0.0379, and 0.0397 kg s_1).Both parallel flow and counter flow configurations were used for all combinations of mass flow rates. The total number of simulations performed was 100 (two tube diameters · four inner flow rates · three shell flow rates · two flow directions). The output of the simulations included the inlet and outlet velocities, mass flow rates, enthalpy rates as well as velocity, pressure and temperature.

Fig.4.1 Geometry and Meshing Domain of parallel flow heat exchanger

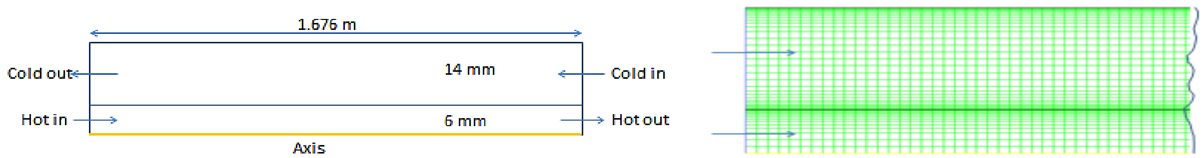
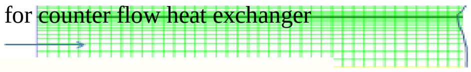


Fig.4.2 Geometry and Meshing domain for counter flow heat exchanger



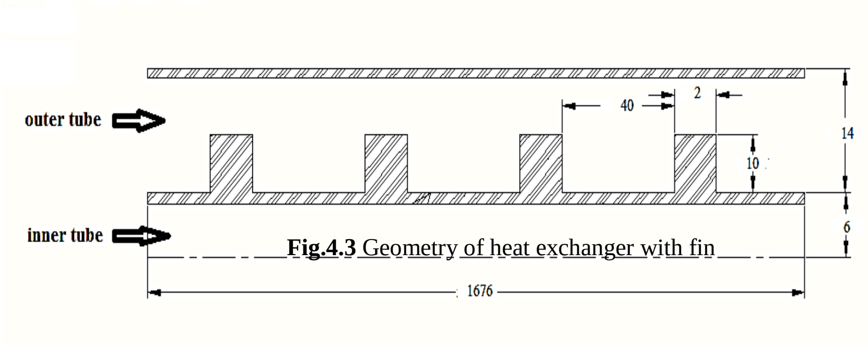


Fig.4.3 Geometry of heat exchanger with fin

Computational Fluid Dynamics (CFD)

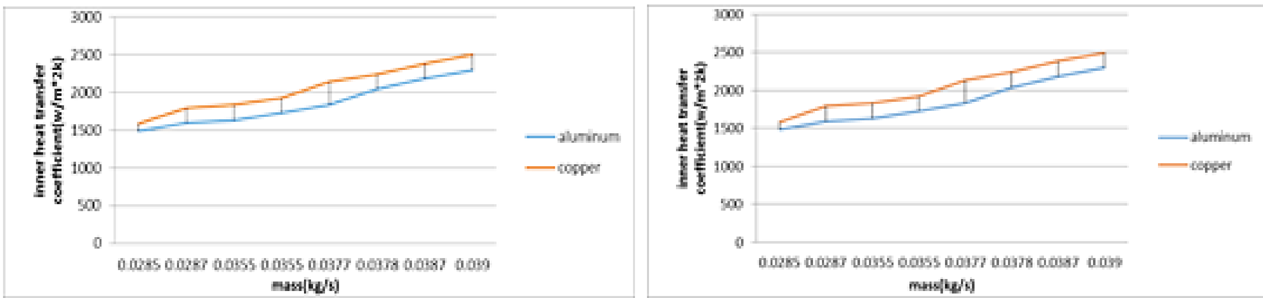
Computational Fluid Dynamics is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. The Physical aspects of any fluid flow are governed by three fundamental principles are A) Mass is conserved B) Newton's second law C) Energy is conserved

These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. Computational Fluid Dynamics (CFD) is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest. During the current investigation, CFD package FLUENT 6.2 was used, which solves CFD problems using finite element method. Steps to be followed when using finite volume method

1. Flow geometry is defined
2. The flow domain is decomposed into a computational mesh or grid – a set of non-overlapping control volumes or cells – over which the integral equations are to be discretized
3. The integral equations are discretized – i.e. approximated in terms of values at a set of nodes
4. The discretized equations are solved numerically
5.

V. RESULT AND DISCUSSIONS

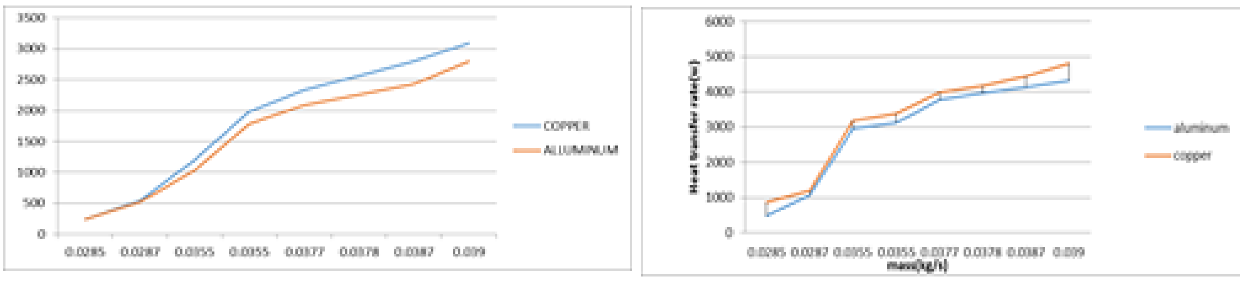
The **Fig. 5.1** shows the inner heat transfer coefficients versus mass flow rate for the case of parallel flow and counter flow. The inner heat transfer rates were based on the inner surface area A1. Heat transfer coefficients were calculated in a similar manner along the length of the tube using the data from each cross-section. This resulted in 32 heat transfer coefficients describing the flow and temperature along the length of the heat exchanger. Both parallel flow and counter flow had inner heat transfer coefficients that were higher difference, for the same given flow rates. The inner heat transfer coefficients increase with increasing mass flow rate of water.



(a) Parallel flow **(b) Counter flow**

Fig.5.1 Represents h_i Vs mass flow rate

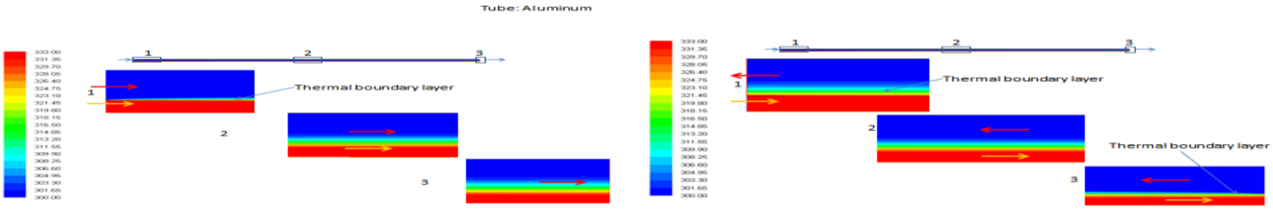
The Fig.5.2 shows the heat transfer rate versus mass flow rate for the case of parallel flow and counter flow. The inner heat transfer rates were based on the surface area of heat exchanger.. Heat transfer rate were calculated in a similar manner along the length of the tube using the data from each cross-section. This resulted in heat transfer rate describing the flow and temperature along the length of the heat exchanger. Both parallel flow and counter flow had heat transfer rate that were nearly lightly difference, for the same given flow rates. The inner heat transfer rate increase with increasing mass flow rate of water.



(a) Parallel flow **(b) Counter flow**

Fig.5.2 heat transfer vs mass flow rate

The following figure represents CFD results for aluminum used as a tube side material –parallel flow & counter flow.



(a)Parallel flow **(b) Counter flow**

Fig. 5.3 Temperature contours

Comparison between the CFD result for various fin size

Validation of heat exchanger through the comparison of theoretical values with CFD results all the results are given above the table from the results theoretical results is almost equal to CFD results here overall heat transfer coefficient, internal heat transfer coefficient, heat transfer rate for parallel flow and counter flow. The overall heat transfer coefficients increase with increasing mass flow rate of water. The ratio of the mass flow rates has a significant effect on the overall heat transfer coefficient, raising the overall heat Transfer coefficient when the flow rate in the shell is increased. When the shell flow rate is high and the inner flow rate is low, the limiting heat transfer is in the inner tube; hence any changes in the inner flow rate will have significant effects on the overall heat transfer rate.

1. Fin having thickness t= 2mm- parallel flow (mass flow rate (m/s) Vs overall heat transfer coefficient w/m^{2}k)

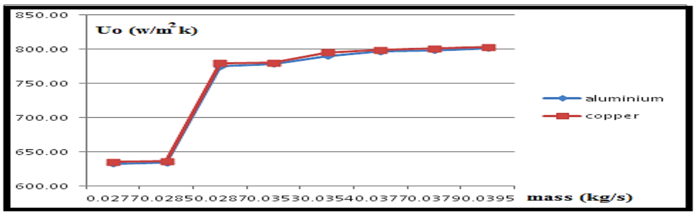


Fig: 5.4 Uo Vs mass flow rate parallel flow

2. Fin having thickness t= 4mm- parallel flow (Mass flow rate (m/s) Vs overall heat transfer coefficient w/m^{2}k)

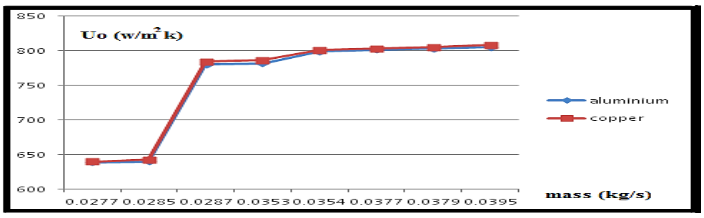


Fig: 5.5 Uo Vs mass flow rate parallel flow

3. Fin having thickness t= 6mm- parallel flow (mass flow rate (m/s) Vs overall heat transfer coefficient w/m²k)

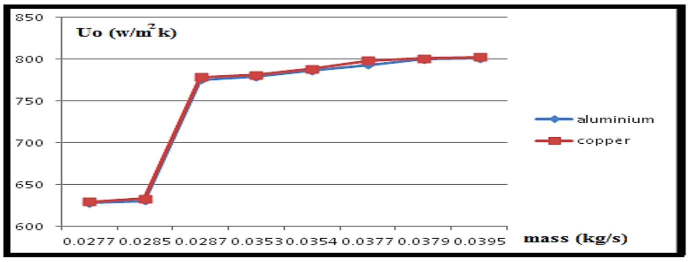


Fig: 5.6 U_O Vs mass flow rate parallel flow

VI. CONCLUSION

A computational fluid dynamics package (Fluent 6.3) was used to study the heat transfer characteristics of a double-pipe heat exchanger under the different parameters such as mass flow rate, flow direction; different materials for both parallel flow and counter flow. Validation runs were performed with the outlet temperature of the cold fluid, hot fluid and velocity. The results of these simulations were well within the range of results from the analytical work. Overall heat transfer coefficients, heat transfer rate, heat transfer coefficient were calculated for parallel and counter flow heat exchanger in the range of 500–2500. The results show an increasing overall heat transfer coefficients as per the mass flow rate. There are some differences in their effects on the heat transfer enhancement factor. Although the heat transfer coefficient is increases as per mass flow rate .This suggests that increasing the inlet velocity of the shell is more effective on enhancing heat transfer than increasing the inlet velocity of the inner tube. In terms of the effects of different flow arrangements on counter flow performs better than Parallel flow. Counter flow provide good heat transfer rate, overall heat transfer coefficient Also providing a fin on its outer surface of the inner tube produces more heat transfer rate as compared with the ordinary tube without fins. Finally, the magnitude of the overall heat transfer coefficient decreases as the total length of the double tube increases.

REFERENCES

a) *ArthurP.Fraas, M.Necati Ozisik, Heat Exchanger Design, John Wiley&Sons, Inc., New York. London. Sydney.*
b) *A.J.Gram,1960 "Mechanical Design of Heat Exchanger," Industrial and Engineering Chemistry, vol.52,p.474.*
c) *D.H.Fax and R.R. Mills, Jr., 1957 "General Optical Heat Exchanger Design ," Trans.ASME, vol.71,p.653.*
d) *Gabriela Huminic, Angel Huminic, 2011, Heat transfer characteristics in double tube helical heat exchangers using Nano fluids Transilvania University of Brasov, Romania.*
e) *H.S.Gardner and I.Siller, 1947 "Shell-Side coefficient of Heat Transfer in a Baffled Heat Exchanger," Trans.ASME, vol.69, P.687.*
f) *Journal of Heat Transfer Estimating no of shells in the shell and tube heat exchanger pp.304-309.*
g) *Ozisik., M.N, 1989, Boundary value problems of Heat conduction, Dovev, New York.*
h) *R.M.Davison, D.W.Rahoi, G.Gemmel, "Stainless steels for heat exchanger service".*
i) *Shyam.N.Singh:1985." Recuperative heat exchanger systems for indirect fired furnaces". W.B. Combustion Inc., Milwaikee. Wisconcin.*
j) *Timothy J. Rennie, Vijaya G.S. Raghavan, 2006, Numerical studies of a double-pipe helical heat exchanger, McGill University, Canada.*
k) *Wen-Lih Chen , Wei-Chen Dung, 2008, Numerical study on heat transfer characteristics of double tube Heat exchangers with alternating horizontal or vertical oval cross section pipes as inner tubes, Taiwan .*
l) *W.Hyrnisak, 1958, Heat Exchanger, academic press.*