"A Review on Basics of Heat Exchanger"

Prof. P.K. Pardakhe[1], Prof. A.B. Samarth[2], Dr. V.L. Bhambere[3], Prof. P.H. Rathod

[1]Assistant Professor at
Jagadambha College of Engineering and Technology Yavatmal Maharastra India
[2]Assistant Professor at
Jagadambha College of Engineering and Technology Yavatmal Maharastra India
[3]Assosiate Professor at
Jagadambha College of Engineering and Technology Yavatmal Maharastra India
[4] Assistant Professor at
Jagadambha College of Engineering and Technology Yavatmal Maharastra India

---***---

Abstract:- *A heat exchanger is equipment that used to transfer or exchange the heat energy for different purposes. In the industries, heat exchangers are very important and play vital role to recover heat between two process fluids. The most common heat transfer devices are concentric tube (double pipe), shell and tube and, plate heat exchanger. Computational Fluid Dynamics (CFD) is an analysis of the physical phenomena involved in fluid flow and heat conduction by computer numerical calculation and graphical display. This chapter discusses the basic design methods for two fluid heat exchangers.*

1.   INTRODUCTION

A heat exchanger is a system used to transfer heat between two or more fluids. Heat exchangers are used in both cooling and heating processes. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air. Another example is the heat sink, which is a passive heat exchanger that transfers the heat generated by an electronic or a mechanical device to a fluid medium, often air or a liquid coolant.

There are three primary classifications of heat exchangers according to their flow arrangement. In parallel-flow heat exchangers, the two fluids enter the exchanger at the same end, and travel in parallel to one another to the other side. In counter-flow heat exchangers the fluids enter the exchanger from opposite ends. The counter current design is the most efficient, in that it can transfer the most heat from the heat (transfer) medium per unit mass due to the fact that the average temperature difference along any unit length is higher. See countercurrent exchange. In a cross-flow heat exchanger, the fluids travel roughly perpendicular to one another through the exchanger.

For efficiency, heat exchangers are designed to maximize the surface area of the wall between the two fluids, while minimizing resistance to fluid flow through the exchanger. The exchanger's performance can also be affected by the addition of fins or corrugations in one or both directions, which increase surface area and may channel fluid flow or induce turbulence.

The driving temperature across the heat transfer surface varies with position, but an appropriate mean temperature can be defined. In most simple systems this is the "log mean temperature difference" (LMTD). Sometimes direct knowledge of the LMTD is not available and the NTU method is used.

2.   Types of heat Exchangers:-

Based on the design characteristics indicated above, there are several different variants of heat exchangers available. Some of the more common variants employed throughout industry include:

1) Shell and tube heat exchangers
2) Double pipe heat exchangers
3) Plate heat exchangers
4) Condensers, evaporators, and boilers

2.1 Shell and Tube Heat Exchangers

A shell and tube heat exchanger is a class of heat exchanger designs. It is the most common type of heat exchanger in oil refineries and other large chemical processes, and is suited for higher-pressure applications. As its name implies, this type of heat exchanger consists of a shell (a large pressure vessel) with a bundle of tubes inside it. One fluid runs through the tubes, and another

fluid flows over the tubes (through the shell) to transfer heat between the two fluids. The set of tubes is called a tube bundle, and may be composed of several types of tubes: plain, longitudinally finned, etc.

Two fluids, of different starting temperatures, flow through the heat exchanger. One flows through the tubes (the tube side) and the other flows outside the tubes but inside the shell (the shell side). Heat is transferred from one fluid to the other through the tube walls, either from tube side to shell side or vice versa. The fluids can be either liquids or gases on either the shell or the tube side. In order to transfer heat efficiently, a large heat transfer area should be used, leading to the use of many tubes. In this way, waste heat can be put to use. This is an efficient way to conserve energy.

The shell and tube exchanger consists of four major parts:

1) Front Header—this is where the fluid enters the tubeside of the exchanger. It is sometimes referred to as the Stationary Header.
2) Rear Header—this is where the tubeside fluid leaves the exchanger or where it is returned to the front header in exchangers with multiple tubeside passes.
3) Tube bundle—this comprises of the tubes, tube sheets, baffles and tie rods etc. to hold the bundle together.
4) Shell—this contains the tube bundle.

Shell and tube heat exchanger design

There can be many variations on the shell and tube design. Typically, the ends of each tube are connected to plenums (sometimes called water boxes) through holes in tubesheets. The tubes may be straight or bent in the shape of a U, called U-tubes
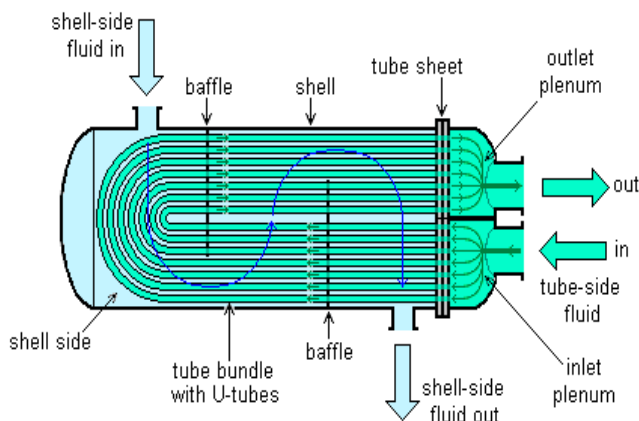
U-tube heat exchanger

Fig. 01 :- U- tube heat exchanger

2.2 Applications and uses

The simple design of a shell and tube heat exchanger makes it an ideal cooling solution for a wide variety of applications. One of the most common applications is the cooling of hydraulic fluid and oil in engines, transmissions and hydraulic power packs. With the right choice of materials they can also be used to cool or heat other mediums, such as swimming pool water or charge air.[5] One of the big advantages of using a shell and tube heat exchanger is that they are often easy to service, particularly with models where a floating tube bundle (where the tube plates are not welded to the outer shell) is available.

Shell and tube Heat Exchangers have the ability to transfer large amounts of heat in relatively low cost, serviceable designs. They can provide large amounts of effective tube surface while minimizing the requirements of floor space, liquid volume and weight.

Shell and tube exchangers are available in a wide range of sizes. They have been used in industry for over 150 years, so the thermal technologies and manufacturing methods are well defined and applied by modern competitive manufacturers.

3. Methodology: -

The basic design methods for two fluid heat exchangers are

1) log-mean temperature difference (LMTD) method

2) the effectiveness ε−NTUmethod

3) dimensionless mean temperature difference (Ψ−P)

4) (P1 – P2) to analyse recuperations.

The LMTD method can be used if inlet temperatures, one of the fluid outlet temperatures, and mass flow rates are known. The ε – NTU method can be used when the outlet temperatures of the fluids are not known. Also, it is discussed effectiveness-modified number of transfer units (ε−NTUo) and reduced length and reduced period (Λ−π) methods for regenerators. (Ψ - P) and (P1 – P2) methods are graphical methods. The (P1 – P2) method includes all major dimensionless heat exchanger parameters. Hence, the solution to the rating and sizing problem is non-iterative straight forward.

3.1. Governing equations

The energy rate balance is

$$dEcvdt = Q^{\cdot} - W^{\cdot} + \sum im^{\cdot}i(hi + Vi22 + gzi) - \sum em^{\cdot}e(he + Ve22 + gze)$$

For a control volume at steady state, $dEcvdt = 0$. Changes in the kinetic and potential energies of the flowing streams from inlet to exit can be ignored. The only work of a control volume enclosing a heat exchanger is flow work, so $W^{\cdot} = 0$ and single-stream (only one inlet and one exit) and from the steady-state form the heat transfer rate becomes simply [1–3]

$$Q^{\cdot} = m^{\cdot}(h2 - h1)E2$$

For single stream, we denote the inlet state by subscript 1 and the exit state by subscript 2.

For hot fluids,

$$Q^{\cdot} = m^{\cdot}(hh1 - hh2)E3$$

For cold fluids,

$$Q^{\cdot} = m^{\cdot}(hc2 - hc1)E4$$

The total heat transfer rate between the fluids can be determined from

$$Q^{\cdot} = UA\Delta TlmE5$$

Nomenclature

$Q^{\cdot}$ = Heat transfer rate, kW

A = Total heat transfer surface area of heat exchanger, total heat transfer surface area of all matrices of a regenerator, m2

ΔTlm = Log-mean temperature difference,°C,K

U = Overall heat transfer coefficient, W/m2K

E = Total energy, kJ

h = Specific enthalpy, kJ/kg

C = Flow stream heat capacity rate, W/K

d, D = Diameter, m

$m^{\cdot}$ = Mass flow rate, kg/s

NTU = Number of transfer units

Greek symbols

B = packing density for a regenerator, m2/m3

Δ = Difference

E = Effectiveness

Pm = Matrix material density, kg/m3

H = Efficiency

Λ = Reduced length for a regenerator

Π = Reduced period for a regenerator

3.3 Computational fluid dynamics (CFD)

Computational fluid dynamics (CFD) analysis is one of the most successful methods of solving familiar heat exchanger problems. HTRI uses CFD simulation effectively in its research and contract projects to improve heat exchanger performance for customers worldwide.

Computational Fluid Dynamics (CFD) is the finite element model technique which is great promise in the study of the fluids flow, heat transfer and etc. Generally finite element model means the model of any body by dividing the body into small parts known as elements. As the body has infinite particles it is impossible to study the action on the each particle hence to avoid this difficulty the body is divided into small elements to get the results. This process is known as the meshing. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modeling starts with describing the boundary and initial conditions for the dominion and leads to modeling of the entire system. CFD is the science of predicting fluid flow pattern, heat and mass transfer, chemical reactions, and related phenomena by solving the set of governing mathematical equations such as conservation of mass, conservation of momentum, conservation of energy, conservation of species, effects of body forces, etc. It is a powerful tool for research and development in heat transfer simulation and modeling of heat exchanger in the multiphase flow systems. However, in these methods are computationally time consuming, obtaining numerical errors during calculation and cannot be predicted all thermal and hydraulic behaviors of fluids in the flow fields. To resolve these difficulties, computational fluid dynamics (CFD) software can be used. Ansys Fluent is feasible and famous software that compute accurately the conjugate heat transfer. Herewith realizable K-Epsilon, standard wall function mode can also be employed. This simulation gives the values of pressure, temperature, heat transfer rate and velocity at various sections of the annulus and pipe. Convection within the fluids (natural and forced), conduction in solid regions, thermal radiation and external heat gain or loss from the outer boundaries of the model can be computed using Ansys Fluent. The performance of parametric analysis on the average value of convective heat transfer coefficient is done based on results of 3D in CFD simulations

characterized by an average flow direction. A significant advantages of CFD methods are evaluating thermal and hydrodynamics of the system that makes possibility to analysis the geometric changes (different feed point layouts such as multiple entry points), selecting operating conditions in lesser time (faster turnaround time). It can be reduced different expenses (flexibility for changing design parameters without hardware changes). Another notable advantage is CFD provides more detailed output information for trouble-shooting and far better understandings of the thermal and hydrodynamics performance.

4. Litrature Review

Gurbir Singh1:- In this paper, the shell and tube heat exchanger is considered in which hot water is flowing inside one tube and cold water runs over that tube. Computational fluid dynamics technique which is a computer based analysis is used to simulate the heat exchanger involving fluid flow, heat transfer. CFD resolve the entire heat exchanger in discrete elements to find the temperature gradients, pressure distribution and velocity vectors. The turbulence model k-ε is used for accurate results from CFD. The temperature variations are calculated from experiment for parallel and counter flow

2) Pintu1, Heat transfer from one fluid to another is important application in engineering science, various type of heat exchanger devices was developed by engineers from previous decades. Research on these devices continues in present era to improve thermal performance. Simple shell and tube heat exchanger was use to validate the experimental results with CFD results. Experimental validation proves that CFD is effective tool for thermal performance investigation. After experimental validation, main research work was focused on effect of latent heat on thermal performance improvement of simple shell and tube heat exchanger using PCM materials with these being discharging in it

3) Vindhya Vasiny Prasad Dubey:- This paper consists of extensive thermal analysis of the effects of severe loading conditions on the performance of the heat exchanger. To serve the purpose a simplified model of shell and tube type heat exchanger has been designed using kern's method to cool the water from 55 to 45 by using water at room temperature. Then we have carried out steady state thermal analysis on ANSYS 14.0 to justify the design. After that the practical working model of the same has been fabricated using the components of the exact dimensions as derived from the designing. We have tested the heat exchanger under various flow conditions using the insulations of aluminium foil, cotton wool, tape, foam, paper etc. We have also tested the heat exchanger under

various ambient temperatures to see its effect on the performance of the heat exchanger. Moreover we have tried to create the turbulence by closing the pump opening and observed its effect on its effectiveness. All these observations along with their discussions have been discussed in detail inside the paper

4) STEPHENRAJ.V Heat exchanger as the name indicates it transfers heat from one fluid to another which are at different temperatures. Heat exchangers are devices built for efficient heat transfer from one fluid to another and are widely used in engineering processes. Some examples are intercoolers, pre-heaters, boilers and condensers in power plants. The heat transfer efficiency depends on both design of heat exchanger and property of working fluid. Some important design parameters such as the pitch ratio, tube length, and tube layer as well as baffle spacing. In this project, the heat transfer efficiency is improved by implementing the full baffle design and travel tube design and analyzing it through CFD flow simulation to find the approximate heat transfer rates. From the simulation results the optimum baffle design and travel tube design for maximum heat transfer rate is identified

5) 1 Sandeep K. Patel A characteristic of heat exchanger design is the procedure of specifying a design. Heat transfer area and pressure drops and checking whether the assumed design satisfies all requirement or not. The purpose of this paper is how to design the shell and tube heat exchanger which is the majority type of liquid –to-liquid heat exchanger. General design considerations and design procedure are also illustrated in this paper. In design calculation HTRI software is used to verify manually calculated result.

6) KARTHIK SILAIPILLAYARPUTHUR The paper considered a review for the design of a shell and tube heat exchanger. Therein, popular analytical techniques such as log mean temperature difference (LMTD) and effectiveness-number of transfer units (ε-NTU) were considered in the analysis. In the design, analysis, performance charts and tables describing the performance of the shell and tube heat exchanger in terms of crucial dimensionless parameters were developed. These fundamental dimensionless parameters account for the thermal & the physical properties of the fluids and the heat exchanger (HX) material. Using the information from the performance charts and tables, a basic design for the shell and tube heat exchanger can be readily formulated.

7) Sumit S. Mukadam1, This study has been undertaken to study the performance of the shell and tube heat exchanger. Shell and tube heat exchangers are one of the most commonly used type of heat exchangers in the industry for heat exchange purpose. This study shows the

effect of various parameters such as pitch layout and baffle spacing on the heat exchanger performance. Standard study the same. The study also shows the simulation work carried out using 'Solidworks 2016' software for the lab as well as industrial heat exchanger From the results of flow simulation of the heat exchanger it can be concluded that the simulation gives results close to those obtained from experimentation.

5. Conclusion

This chapter has discussed the basic design methods for twofluid heat exchangers. The design techniques of recuperators and regenerators.

Referance: -

1] Temperature Analysis of Shell and Tube Heat Exchanger International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 05 Issue: 04 | Apr-2018

2] The Design of Shell and Tube Heat Exchangers – A Review International Journal of Mechanical and Production Engineering Research and Development (IJMPERD) ISSN (P): 2249-6890; ISSN (E): 2249-8001 Vol. 9, Issue 1, Feb 2019, 87-102

analytical calculations are used to

3] Design and Analysis of Heat Exchanger for Maximum Heattransfer Rate (Multimodel Optimisation Technique) International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 05 Issue: 01 | Jan-2018

4] Shell & Tube Heat Exchanger Thermal Design With Optimization of Mass Flow Rate and Baffle Spacing International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974

5]Performance Analysis of Shell & Tube Type Heat Exchanger under the Effect of Varied Operating Conditions IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 3 Ver. VI (May- Jun. 2014

6] CFD based study of thermal performance of shell and tube heat exchanger with discharging PCM International Journal of Latest Engineering Research and Applications (IJLERA) ISSN: 2455-7137 Volume – 02, Issue – 11, November – 2017, PP – 93-100