Research and Applications of Thermal Engineering
Volume 6 Issue 3
DOI: [To be assigned]

A Case Study on Basic of Heat Exchanger

Walsh Christian*, RonakRana, Hiren Rathod, Dhiraj Patel, Atul Patel
Assistant Professor, Department of Mechanical Engineering, Sardar Patel College of Engineering, Bakrol, Indi

****Corresponding Author***
E-Mail Id:christianwalsh1995@gmail.com

ABSTRACT
Equipment that exchanges or transfers heat energy for various uses is called a heat exchanger. Heat exchangers are essential in the industrial sector because they recover heat between two process fluids. The most popular heat transfer devices are plate heat exchangers, shell and tube heat exchangers, and concentric tube (double pipe).The study of fluid flow and heat conduction by computer numerical calculation and graphical display is known as computational fluid dynamics, or CFD. The fundamental design techniques for two fluid heat exchangers are covered in this chapter.

Keywords: *Heat exchanger, fluid, CFD*

INTRODUCTION

A device that transfers heat between two or more fluids is called a heat exchanger. Heat exchangers are employed in the heating and cooling of systems. To avoid mixing, the fluids can be in direct touch or separated by a solid wall. They are extensively utilised in sewage treatment, chemical plants, petrochemical facilities, power plants, natural gas processing, refrigeration, air conditioning, and space heating. An internal combustion engine is the classic example of a heat exchanger. In this engine, air passes by radiator coils and a circulating fluid called engine coolant cools the coolant while heating the incoming air. The heat sink, a passive heat exchanger that transfers the heat, is another example. Heat released into a fluid medium—typically air or a liquid coolant—by a mechanical or electrical device. Based on how their flows are arranged, heat exchangers can be divided into three main categories. The two fluids enter the heat exchanger at the same end and move parallel to one another to the other in a parallel-flow heat exchanger. The fluids entering counter-flow heat exchangers do so from opposite directions. Heat exchangers are made to be as efficient as possible by optimising the walls surface area between the two fluids and reducing flow resistance. The inclusion of fins or corrugations in one or both directions, which increase surface area and may channel fluid flow or produce turbulence, can also have an impact on the exchanger's performance. Position affects the driving temperature over the heat transfer surface, yet a suitable means temperature can be determined. This is the "log mean temperature difference" (LMTD) in the majority of basic systems. The NTU technique is employed when direct information of the LMTD is unavailable.[1,2]

TYPES OF HEAT EXCHANGER

A variety of heat exchanger variations are available, based on the design features mentioned above. Several of the more widely used variations in the industry are as follows:

- Heat exchangers with shell and tube

- Heat exchangers with two pipes
- Heat exchangers on plates
- Boilers, evaporators, and condensers

SHELL AND TUBE HEAT EXCHANGERS

A shell and tube heat exchanger is a class of heat. 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.

Through the heat exchanger flow two fluids with initial temperatures that are not the same. One travels within the shell (the shell side), while the other travels outside the tubes (the tube side). Through the tube walls, heat is transmitted from one fluid to another, either from the tube side to the shell side or the other way around. On the shell or tube side, the fluids might be either liquids or gases .A vast heat transfer area necessitates the employment of several tubes in order to transfer heat efficiently. Waste heat can be utilized in this way. This is a productive method of energy conservation. The shell and tube exchanger consists of our major parts:[6,7]

- Front Header—this is where the fluid enters the tube side of the exchanger. It is sometimes referred to as the Stationary Header.
- Rear Header—this is where the tube side fluid leaves the exchanger or where it is returned to the front header in exchangers with multiple tube side passes.
- The tubes, tube sheets, baffles, tie rods, and other components that hold the bundle together make up the tube bundle.

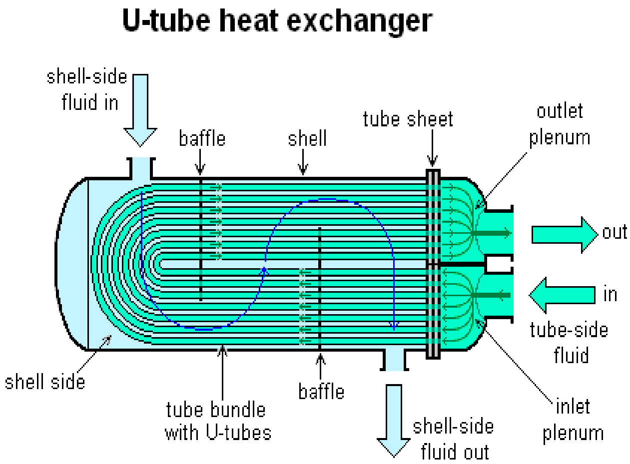

***Fig. 1:** U-tube heat exchanger.*

The shell and tube design can take many different forms. Typically, the ends of each tube are connected to plenums (also known as water boxes) via perforations in tube sheets. The tubes can be straight or twisted into the shape of a U, which is known as U-tubes.

APPLICATIONS AND USES

The simple design of the shell and tube heat exchanger makes it an ideal cooling solution for many applications. One of the most common applications is the cooling of hydraulic fluid and oil in engines, gearboxes and hydraulic units. With properly chosen materials, they can also be

Research and Applications of Thermal Engineering
Volume 6 Issue 3
DOI: [To be assigned]

used to cool or heat other media such as pool water or air charging. [4,5] One of the great advantages of using a shell and tube heat exchanger is that they are often easy to maintain, especially on models available with a floating tube bundle (where the tube plates are not welded to the outer shell).

Shell and tube heat exchangers have the ability to transfer large amounts of heat with a relatively inexpensive and maintainable structure. They can provide a large amount of effective pipe area while minimizing floor space, fluid volume and weight. Shell and tube exchangers are available in a wide range of sizes. They have been used in industry for more than 150 years, so thermal technologies and production methods are well defined and implemented by modern competitive manufacturers.

METHODOLOGY

The basic design methods for two fluid heat exchangers are

- Log-mean temperature difference (LMTD) method
- The effectiveness ε−NTU method
- Dimensionless mean temperature difference(Ψ−P)
- (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. In situations where the fluids' exit temperature is unknown, the ε-NTU approach may be employed. Additionally, it covered the efficiency of modifying the number of transfer units (ε−NTU) and reducing the length and period (λ−π) of regenerator methods. Graphical techniques are the (Ψ - P) and (P1 - P2) methods. Every significant dimensionless heat exchanger parameter is included in the (P1–P2) approach. Thus, the solution to the problem of scaling and the ring is not a simple, iterative process.

GOVERNING EQUATIONS

dE cvdt=Q˙−W˙+∑ im˙I (hi+Vi22+gzi) −∑em˙e (he+Ve22+gze) is the energy rate balance. dEcvdt=0 for a steady-state control volume. One can ignore changes in the kinetic and potential energy of the streams as they pass from the intake to the exit. The sole labor of a control volume around a heat exchanger is flow work, meaning W˙=0 and single-stream (having just one input and one outlet), and the heat transfer rate becomes straightforward from the steady-state form[1-3].

Q˙=m˙(h2−h1)E2

For a single stream, we indicate the entrance state with subscript 1 and the exit state with subscript 2.
In the case of heated fluids,

Q = m (hh1 hh2).E3

In the case of cold fluids,

Q = m (hc2 hc1).E4

The total heat transfer rate between the fluids can be calculated using

Q˙=UA ΔT lm E5

Nomenclature

Research and Applications of Thermal Engineering
Volume 6 Issue 3
DOI: [To be assigned]

$\dot{Q}$=Heat transfer rate, kW

A=is equal to the sum of the heat transfer surface areas of the heat exchanger and each matrix in the regenerator, measured in m^2.

ΔT lm=Log-mean temperature difference, °C ,
KU = Overall heat transfer coefficient, W/m2 KE=Total energy, kJ
h=Specific enthalpy, kJ/kg

C=Flow stream heat capacity rate, W/Kd,D= Diameter,m
$\dot{m}$=Mass flow rate, kg/s

NTU=Number of transfer units Greek symbols
B=packingdensityforaregenerator,m2/m3Δ=Difference

Λ = Reduced length for a regenerator Π=Reduced period for a regenerator

COMPUTATIONAL FLUID DYNAMICS (CFD)

Known heat exchanger problems can be most successfully solved using computational fluid dynamics (CFD). HTRI efficiently enhances heat exchanger performance for clients worldwide through research and contract projects that make use of CFD simulation. Computational Fluid Dynamics (CFD) is a finite element modeling technique that holds great promise in the study of fluid flow, heat transfer, etc. In general, a finite element model means a finite element**,** anybody 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 an area into small volumes where the equations are solved by repeated methods.

Modeling begins with the description of the Dominion boundary and initial conditions and leads to the 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 governing mathematical equations such as conservation of mass, conservation of momentum, conservation of energy, conservation of species, body forces, etc. It is a powerful tool for heat transfer research and development in simulation and heat exchanger modeling in multiphase flow systems. However, these methods are computationally time-consuming, contain calculation errors, and cannot predict the thermal and hydraulic behavior of all fluids in flow fields. CFD software can be used to solve these problems. ANSYS Fluent is viable and famous software that calculates conjugate heat transfer accurately. In this case, the standard K-Epsilon wall function mode can also be used. This simulation provides values of pressure, temperature, heat transfer and velocity in different parts of the annulus and tube. ANSYS Fluent can be used to calculate convection in fluids (natural and forced), conduction in solid regions, thermal radiation, and external heat gain or loss from the external boundaries of the model. To perform a parametric analysis of the average value of the convective heat transfer coefficient is based on the 3D results of CFD simulations characterized by an average flow direction

A significant advantages of CFD methods are evaluating thermal and hydro dynamics of the system that makes possibility 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 the changes).Another notable advantage is CFD provides more detailed output information for trouble-shooting and far better understandings of the thermal and hydrodynamics performance.

LITRATURE REVIEW

Gurbir Singh

The shell and tube heat exchanger, in which hot water flows through one tube and cold water runs over it, is discussed in this paper. The computer-based technique known as computational fluid dynamics is used to simulate heat transfer in fluid flow heat exchangers. To decide the temperature inclinations, pressure dissemination, and speed vectors, CFD settle the total intensity exchanger into discrete parts. For more precise CFD results, the disturbance model k-ε is used. Experiments are used to calculate temperature changes for parallel and counter flow.

Pintu

The transmission of heat from one fluid to another is a significant application in engineering science. Throughout the years, engineers have invented a variety of heat exchanger systems. In the modern day, research on these devices is ongoing to enhance thermal performance. A basic shell and tube heat exchanger was used to compare the CFD results with the experimental findings. Experimental validation is a useful method for investigating thermal performance in a CFD. Following experimental validation, the primary study emphasis was on how latent heat affected the thermal performance enhancement of a basic shell and tube heat exchanger employing PCM materials that were discharging in it.

Dubey, Vindhya Vasiny Prasad

This research offers a thorough thermal analysis of the impact of extreme loading circumstances on heat exchanger performance. A simpler form of a shell and tube type heat exchanger has been constructed to chill the water from 55 to 45 degrees Celsius using water at ambient 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 aluminum 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. Furthermore, we attempted to produce turbulence by closing the pump opening and observing the effect on its effectiveness. All of these observations, as well as their debates have been thoroughly covered within the study.

Stephenraj V. Heat Exchanger

As the name suggests, it transmits heat from one fluid at one temperature to another. Heat exchangers are frequently utilized in engineering processes because they are designed to transfer heat from one fluid to another efficiently. Power plant condensers, boilers, pre-heaters, and intercoolers are a few examples. The working fluid's properties and the heat exchanger's design both affect heat transfer efficiency.

CONCULSION

The fundamental design techniques for two-flow heat exchangers are covered in

Research and Applications of Thermal Engineering
Volume 6 Issue 3
DOI: [To be assigned]

this chapter. Methods of recuperate and regenerator design.

REFERENCE

1. Sekulic, D. P. (2018). Compact heat exchangers. In *Handbook of thermal science and engineering* (pp. 1501-1520). Springer, Cham.
2. Nptel. (2006). Lecture 1: Heat Exchangers Classifications. Chem. Eng. Des. - II.
3. Horvath. (1983). Heat exchanger design. Glas. Int.
4. Thulukkanam, K. (2000). *Heat exchanger design handbook*. CRC press.
5. McDonald, A. G., & Magande, H. (2012). *Introduction to thermo-fluids systems design*. John Wiley & Sons.
6. Chen, P. (2013). Data Model
7. Conceptual Model-The basic concept of Er model. 2(3), 1–10.
8. What is a Heat Exchanger? http://www.thermex.co.uk/news/blog/160-what-is-a-heat-exchanger.
9. Liu, W., Liu, Z., Wang, Y., & Huang, S. (2009). Flow mechanism and heat transfer enhancement in longitudinal-flow tube bundle of shell-and-tube heat exchanger. *Science in China Series E: Technological Sciences*, *52*(10), 2952-2959.

Cite as: