DESIGN AND DEVELOPMENT OF SHELL AND TUBE HEAT EXCHANGER BY USING CFD

Harshwardhan Uddhage*, Sanjay Kumbhare, Krishna Kumar Thakur*****

*M. Tech RS, Dept. of Thermal Engg., PCST, Bhopal, harshwardhan.uddhage@gmail.com
**Professor, Dept. of Thermal Engg., PCST, Bhopal, sanjaykumbhare2709@gmail.com
***Professor, Dept. of Thermal Engg., PCST, Bhopal, krishnakumarthakur@gmail.com

ABSTRACT

Heat exchanger is a device used to transfer heat from one fluid to another fluid either in direct contact with each other or separated by solid wall. In heat exchangers, there are usually no external heat and work interactions. Typical applications involve heating or cooling of a fluid stream of concern and evaporation or condensation of single- or multi-component fluid streams. In other applications, the objective may be to recover or reject heat, or sterilize, pasteurize, fractionate, distill, concentrate, crystallize, or control a process fluid. In a few heat exchangers, the fluids exchanging heat are in direct contact. In most heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transient manner. In many heat exchangers, the fluids are separated by a heat transfer surface, and ideally they do not mix or leak. Such exchangers are referred to as direct transfer type, or simply recuperators. In contrast, exchangers in which there is intermittent heat exchange between the hot and cold fluids—via thermal energy storage and release through the exchanger surface or matrix— are referred to as indirect transfer type, or simply regenerators. Such exchangers usually have fluid leakage from one fluid stream to the other, due to pressure differences and matrix rotation/valve switching. Commercially, heat exchangers are known as boilers, condensers, air heaters, cooling towers in power industry, radiator in automobile industry, equipments in chemical industry.

Recuperators are further sub classified as prime surface exchangers and extended-surface exchangers. Prime surface exchangers do not employ fins or extended surfaces on any fluid side. Plain tubular exchangers, shell-and-tube exchangers with plain tubes, and plate exchangers are good examples of prime surface exchangers.

NOMENCLATURE

Notation	Description	Unit
A_{bm}	Mean of interior cross flow area and window zone flow area	m^2
A_{sc}	Interior cross flow section area at or near the shell centerline	m^2
A_{wz}	Window zone flow area	m^2
B_c	Baffle cut percentage	-
B_s	Interior sections baffle spacing	m
B_{si}	Inlet section baffle spacing	m
B_{so}	Outlet section baffle spacing	m
C_f	Factor used in friction factor	-
D_b	Baffle diameter	m
D_c	Diameter of circle passing through center of outermost tubes in shell	m
D_0	Tube bundle diameter	m
D_s	Shell inside diameter	m
d_0	Tube outside diameter	m
f	Fanning friction coefficient	-
f_b	Correction factor for bundle bypass stream for pressure drop	-
f_1	Correction factor for effect of baffle leakage for pressure drop	-
f_s	Correction factor for unequal baffle spacing at end sections	-
h_{id}	Ideal heat transfer coefficient for pure cross flow in ideal tube bank	W/m^2K
J_b	Correction factor for bundle and pass partition bypass stream	-
J_b	Correction factor for bundle and pass partition bypass stream	-
J_c	Correction factor baffle configuration	-
J_l	Correction factor baffle leakages effect, including both baffle to shell and baffle to tubes	-
J_r	Correction factor for any adverse temperature gradient build up in	-

ISSN 2250-3153

	laminar flows	
J_s	Correction factor for large baffle spacing at the inlet and outlet sections compared to the central baffle spacing	-
L	Vertical height of cross flow zone	m
l	Distance between end plate tubes	m
N_b	Number of baffles	-
N_c	Number of tube rows in cross flow	-
N_w	Number of tube rows in window section	-

1. INTRODUCTION

Shell-and-tube heat exchangers are the most versatile type of heat exchanger. They are used in many industrial areas, such as power plant, chemical engineering, petroleum refining, petrochemicals industries, food processing, paper industries, etc. Shell and tube heat exchangers provide relatively large ratios of heat transfer area to volume and weight and they can be easily cleaned. They have greater flexibility to meet any service requirement. Shell and tube heat exchangers can be designed for high pressures relative to the environment and high pressure differences between fluid streams [2].

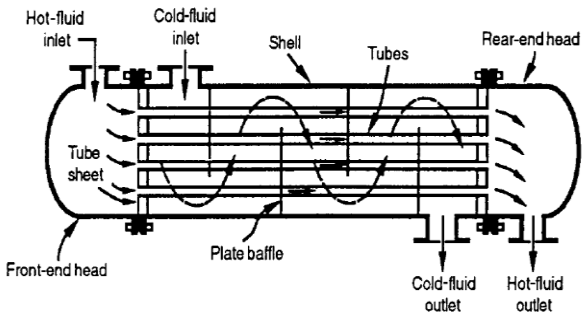


Figure 1.1 Shell-and-tube exchanger (BEM) with one shell pass and one tube pass or Fixed-tube sheet heat exchanger.

Shell and tube heat exchanger consist of bundle of round tubes mounted in cylindrical shell with tubes parallel to shell. One fluid flows through tubes, while another fluid flows across and along the axis of the exchanger. Major components of shell and tube heat exchanger are shell, Tubes, baffles, front-end head, rear-end head and tube sheets as shown in figure 1.1.

A variety of different internal constructions are used in shell-and-tube exchangers, depending on the desired heat transfer and pressure drop performance and the methods employed to reduce thermal stresses, to prevent leakages, to provide for ease of cleaning, to contain operating pressures and temperatures, to control corrosion, to accommodate highly asymmetric flows, and so on. Various Front end stationary head & rear end head types and shell types have been standardized by TEMA (Tubular Exchanger Manufactures Association). They are identified by an alphabetic character, as shown in figure 1.2. TEMA has developed a notation system to designate major types of shell-and-tube exchangers. In this system, each exchanger is designated by a three-letter combination, the first letter indicating the front-end head type, the second the shell type, and the third the rear-end head type. Some common shell-and-tube exchangers are AES, BEM, AEP, CFU, AKT, and AJW

Out of the all heat exchanger, E-shell is the most common due to its cheapness and simplicity. In this shell, the shell fluid enters at one end of the shell and leaves at the other end, i.e., there is one pass on the shell side. The tubes may have a single or multiple passes and are supported by transverse baffles. This shell is the most common for single phase shell fluid application. With a single tube pass, a nominal counter flow can be obtained.

Tubes are classified as U-tube configuration and fixed tube sheet configuration. U-tube configuration allows independent expansion of tubes and shell. Therefore, thermal expansion is unlimited. The U-tube, shown in figure 1.3, is the least expensive construction because only one tube sheet is needed. The tube side cannot be cleaned by mechanical means because of the U-bend. Only an even number of tube passes can be accommodated. Individual tubes cannot be replaced except in the outer row. A fixed tube sheet configuration is shown in figure 1.1. The shell is welded to the tube sheets and there is no access to the outside of the tube bundle for cleaning. The low cost option has only limited thermal expansion, which can be somewhat increased by expansion bellows. Individual tubes are replaceable. Cleaning of the tubes is mechanically easy [1& 2].

Also, tube bundle layout is characterized by the included angle between tubes, as shown in figure 1.4. A layout of 30° results in the greatest tube density but for external cleaning purpose, a square 90° or 45° layout is suitable. Tube pitch, P_T, is usually chosen so that the pitch ratio P_T/d_o, is between 1.25 and 1.5. When tubes are too close, the tube sheet becomes structurally weak .

Baffles perform three major functions: their main function is to support the tubes for structural rigidity, preventing tube vibration and sagging, secondly, to maintain the velocity of shell side fluid flow. And third is to increase the heat transferred between the fluids. And this is accomplished by two ways. Firstly, they increase the residence time of the shell side fluid by making it to flow in a zig-zag, path. This increases the available time for the process of heat transfer. This effect increases the total heat transferred. Secondly baffles create extra turbulence and increase the shell side heat transfer coefficient. By adjusting the baffle openings (known as baffle cut) and baffle spacing, it is possible to vary the heat transfer rates. By using the proper baffles, the flow dead points in the shell can also be removed. As per the TEMA standard, minimum baffle spacing should be one fifth of shell inside diameter or 2 inch whichever is greater. Closer baffles spacing affect to the mechanically cleaning of outside tube bundle and it also results in poor stream distribution. For single-phase fluids on the shell side, a horizontal baffle cut (Figure 10) is recommended, because this minimizes accumulation of deposits at the bottom of the shell and also prevents stratification. However, in the case of a two-pass shell (TEMA F), a vertical cut is preferred for ease of fabrication and bundle assembly. Also, both very small and very large baffle cuts are detrimental to efficient heat transfer on the shell side as they create vortices inside the shell. And baffle cuts ranging from 20% to 36% are considered as very efficient [2].

Baffles may be classified as transverse and longitudinal types (for example, the F-shell has a longitudinal baffle). The transverse baffle may be classified as plate baffles and rod baffles. The most commonly used Plate baffles may be single-segmental, double-segmental, or triple-segmental, as shown in Figure 1.5.

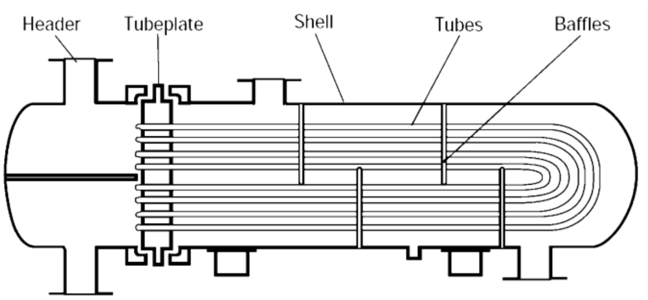


Figure 1.2 U-tube heat exchanger

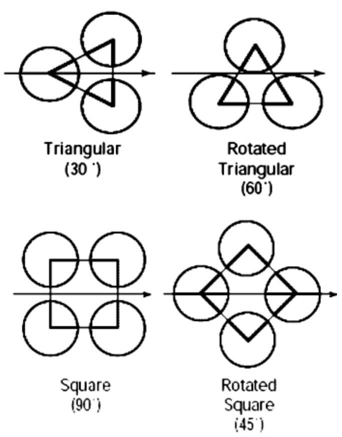
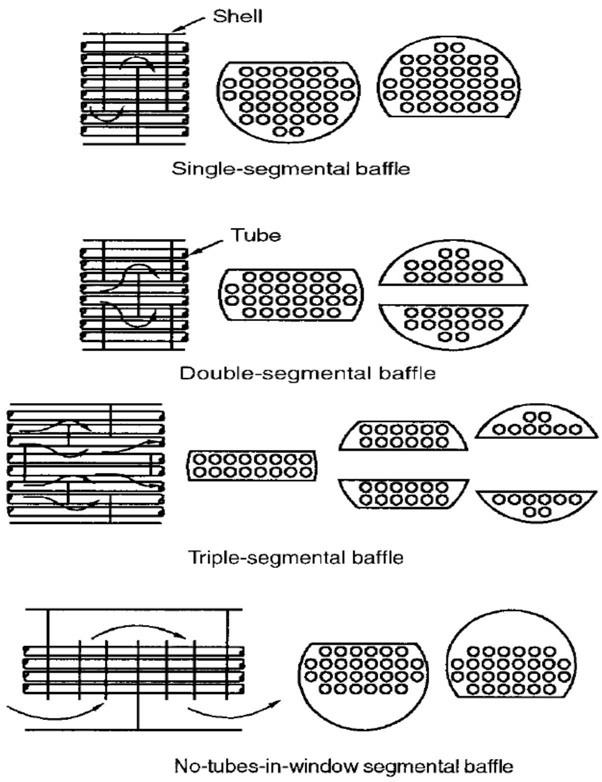


Figure 1.3 Tube Layout Angles



The main considerations taken into account in the design of heat exchanger for a particular application are thermal analysis, mechanical design, pressure drop characteristics, design for manufacture and physical size and cost. A selected shell and tube heat exchanger must satisfy the process requirements with the allowable pressure drops until the next scheduled cleaning of the plant. The basic logical structure of the process heat exchanger design procedure [2] is shown in figure 1.6.

1.2 NECESSITY

Heat exchangers have always been an important part to the life cycle and operation of many systems. A heat exchanger is a device built for efficient heat transfer from one medium to another in order to carry and process energy. It causes a large shell-side pressure drop due to the sudden contraction and expansion of the flow in the shell side, and the fluid impinging on the shell walls caused by segmental baffles. It results in a dead zone in each compartment between two adjacent segmental baffles, leading to an increase of fouling resistance and causes low heat transfer efficiency.

The design of a heat exchanger requires a balanced approach between the thermal design and pressure drop. The pressure drop results in the increase of the operating cost of fluid moving devices such as pumps and fans. This shows that along with the design for the capacity for heat transfer, the pressure drop determinations across the heat exchanger are equally important. Also it is necessary to develop the general purpose heat exchanger which would be suitable for different fluid properties.

1.3 OBJECTIVE

The objectives of this work are as follows

1. To study the fluid flow and heat transfer characteristics of shell and tube heat exchanger.
2. To optimize the heat exchanger for better performance.
3. To reduce the pressure drop & improve heat transfer characteristics.
4. And to develop the heat exchanger which would be applicable for different fluid properties.

2. COMPUTATIONAL FLUID DYNAMICS

2.1 INTRODUCTION

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer (not for analysis of chemical reactions). Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products.

It can be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimise equipment are very inexpensive with CFD when compared to experiments.

2.2 CFD COMPUTATIONAL TOOLS

There is a variety of commercial CFD software available such as Fluent, Ansys CFX, STAR-CD, as well as a wide range of suitable hardware and associated costs, depending on the complexity of the mesh and size of the calculations. The work for this project was solved on an IBM workstation with Pentium 4 processors totalling 32 GHz RAM, running on Linux Operating System. In this project ICEM CFD is used to mesh the model and CFX is used for Pre-Processing, solving and Post-Processing of simulation results.

To run a simulation, three main elements are needed:

1. **Pre-processor:** A pre-processor is used to define the geometry for the computational domain of interest and generate the mesh of control volumes (for calculations). Generally, the finer the mesh in the areas of large changes, the more accurate the solution. Fineness of the grid also determines the computer hardware and calculation time needed.
2. **Solver:** The solver makes the calculations using a numerical solution technique, which can use finite difference, finite element, or spectral methods. Most CFD codes use finite volumes, which is a special finite difference method. First the fluid flow equations are integrated over the control volumes (resulting in the exact conservation of relevant properties for each finite volume), then these integral equations are discretized (producing algebraic equations through converting of the integral fluid flow equations), and finally an iterative method is used to solve the algebraic equations
3. **Post-Processor**: The post-processor provides for visualisation of the results, and includes the capability to display the geometry/mesh, create vector, contour, and 2D and 3D surface plots. Particles can be tracked throughout a simulation, and the model can be manipulated (i.e. changed by scaling, rotating, etc.), and all in full colour animated graphics.

2.3 PROBLEM SOLVING WITH CFD

There are many decisions to be made before setting up the problem in the CFD code. Some of the decisions to be made can include: whether the problem should be 2D or 3D, which type of boundary conditions to use, whether or not to calculate pressure/temperature variations based on the air flow density, which turbulence model to use, etc. The assumptions made should be reduced to a level as simple as possible, yet still retaining the most important features of the problem to be solved in order to reach an accurate solution. After the above decisions are made, the geometry and mesh can be created. The grid should be made as fine as required to make the simulation ‘grid independent’. To determine the fineness required, a grid dependence study is normally carried out by making a series of refinements on an initially course grid, and carrying out simulations on each to determine when the key results of interest do not change, at which point the grid is considered independent.

3. CONCLUSION

In this analysis, numerical simulation for heat exchanger with different number of baffles, baffle cut, tube diameter and tube length are performed to reveal the effect of different baffle configuration on heat transfer and pressure drop characteristics. Also the effect of fin on heat transfer characteristics and pressure drop of heat exchanger are performed. The major findings are summarized as follows:

1. The increase in number of baffles leads to turbulence of fluid flow which causes increase in heat transfer

characteristics but also leads to increase in pressure drop.

2. The flow profile of main fluid stream depends upon the number of baffles, their arrangement, height of baffle cut and tube length. In certain rage, the number of baffles, shortening the height of baffles and length of tube can decrease the pressure drop as well as increase the heat transfer coefficient effectively.
3. Also the diameter of tube influences the heat transfer characteristics. Since the surface area of smaller tubes is lesser, the heat transfer rate is also lower. Hence suitable diameter is to be chosen to increase the heat transfer rate.
4. The tubes with fin occupy more space inside the shell. Hence pressure drop of heat exchanger is higher than the normal heat exchanger. Also surface area of finned tube is larger which increase the heat transfer rate but reduces the heat transfer coefficient. Thus finned tubes are to be incorporated where higher heat transfer rate is of primary importance.
5. In this analysis, the new heat exchanger with reduced tube length with 10 baffles and 36% baffle cut shows the best performance. The length of such heat exchanger is kept as 1540 mm, baffle spacing is kept as 0.14 m and other dimensions are kept
6. In case heat exchanger with finned tube, the heat exchanger with 10 baffles and 36% baffle cut shows the best performance. For such heat exchanger the middle section and inlet & outlet section baffle spacing is kept as 0.165 m and 0.1675 m respectively. The other dimensions are kept same as

REFERENCES

[1] S. C. Arora, S. Domkundwar and A. V. Domkundwar, “A course in Heat and Mass Transfer”, Dhanpat Rai and company, Sixth Edition, 2000.

[2] Sadik Kakac, Hongtan Liu., “Heat Exchangers selection, rating and thermal design”, *CRC Press LLC*, Second Edition, 2002.

[3] Ender Ozden, Ilker Tari, “Shell side CFD analysis of a small shell-and-tube heat exchanger”, *Energy Conversion and Management*, Vol- 51 (2010), pp: 1004–1014.

[4] Uday C. Kapale, Satish Chand, “Modeling for shell-side pressure drop for liquid flow in shell-and-tube heat exchanger”, *International Journal of Heat and Mass Transfer*, Vol- 49 (2006), pp: 601–610.

[5] Yan Li, Xiumin Jiang, Xiangyong Huang, Jigang Jia, Jianhui Tong, “Optimization of high-pressure shell-and-tube heat exchanger for syngas cooling in an IGCC”, *International Journal of Heat and Mass Transfer,* Vol-53 (2010), pp: 4543–4551.

[6] Apu Roy, D.H.Das, “CFD analysis of a shell and finned tube heat exchanger for