Journal of Engineering Research

Volume 6 | Issue 3 Article 17

2022

A Review on Comparative Performance of Shell-and-tube Heat Exchangers with Various Design Configurations

Heba Abdelhamid

Recommended Citation

Abdelhamid, Heba (2022) "A Review on Comparative Performance of Shell-and-tube Heat Exchangers with Various Design Configurations," *Journal of Engineering Research*: Vol. 6: Iss. 3, Article 17.
Available at: https://digitalcommons.aaru.edu.jo/erjeng/vol6/iss3/17

Vol. 6 – No. 3, 2022 Journal of Engineering Research (ERJ)

A Review on Comparative Performance of Shell-and-tube Heat Exchangers with Various Design Configurations

Heba Roushdy Mohamed Abdelhamid, Ayman Ibrahim Bakry, Hagar Alm ElDin Mohamad
Department of Mechanical Engineering, Faculty of Engineering, Tanta University, Tanta, Egypt
Emails: heba147824@f-eng.tanta.edu.eg , hagaralmeldin@f-eng.tanta.edu.eg

***Abstract*- Widely known that, heat exchanger is a device that is being used to transfer thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. The most common heat transfer devices are concentric tube (double pipe), shell and tube and, plate heat exchanger. Nowadays, Shell and tube heat exchangers were used extensively in most industries as petrochemicals, oil and refineries. According to previous survey, almost 45% of heat exchangers utilized are shell and tube heat exchangers due to its high-pressure application it is more suitable in the field of oil & petrochemical application. Thus, the following study presented shell and tube type only. Due to complexity of studying heat exchangers experimentally, Computational Fluid Dynamics (CFD) used to simulate the effect of local surface heat transfer coefficients on the surfaces by aid of computer numerical calculation and graphical display. In addition, analysis of the physical phenomena involved in fluid flow and heat conduction would be presented in the following research paper with Comparative designs for shell and tube heat exchangers. The paper considered a review for the design of a shell and tube heat exchanger. A variety of heat exchangers are used in industry and in their products. The objective of this paper is to describe most of these heat exchangers in some detail using classification schemes and the basic design methods for two fluid heat exchangers. The design techniques of recuperators and regenerators. 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 case considered herein, both LMTD and ε-NTU techniques yield the same exact results.**

Keywords- Heat Exchanger; Shell and tube heat Exchanger; parallel flow; counter flow; single pass shell and tube.

I. INTRODUCTION

A heat exchanger is a system used to transfer heat between two or more fluids. In heat exchangers, there are usually, no external heat and work interactions. 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.

Typical applications involve heating or cooling of a fluid stream of concern and evaporation or condensation of single or multicomponent fluid streams. In many heat exchangers, the fluids are separated by a heat transfer surface, and ideally, they do not mix or leak. 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 most well-known type of heat exchanger is a car radiator. In a radiator, a solution of water and ethylene glycol, also known as antifreeze, transfers heat from the engine to the radiator and then from the radiator to the ambient air flowing through it. This process helps to keep a car's engine from overheating.[1] 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.

Types of heat Exchangers: -

Based on the design characteristics shown in the following in figures illustrated below from figure1:6, there are different types of heat exchangers existing.

Some of the more common types used throughout industry include:

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

Classifications of heat exchangers:

In the following sections, we will introduce more details about each of Heat exchanger types.

A. *Shell and tube heat exchangers*

Shell and Tube Heat Exchangers are one of the most popular types of heat exchangers due to the flexibility. The shell and tube heat exchangers design allow for a wide range of pressures and temperatures. There are two main categories of Shell and Tube exchanger:[2]

A shell-and-tube exchanger consists of a number of tubes mounted inside a cylindrical shell which is a typical unit that may be found in a petrochemical plant. Two fluids can exchange heat, (Heat is transferred from one fluid to the other through the tube walls, either from tube side to shell side or vice versa) one fluid flows over the outside of the tubes while the second fluid flows through the tubes. The fluids can be single or two phase and can flow in a parallel or a cross/counter flow arrangement. 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 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). [3]

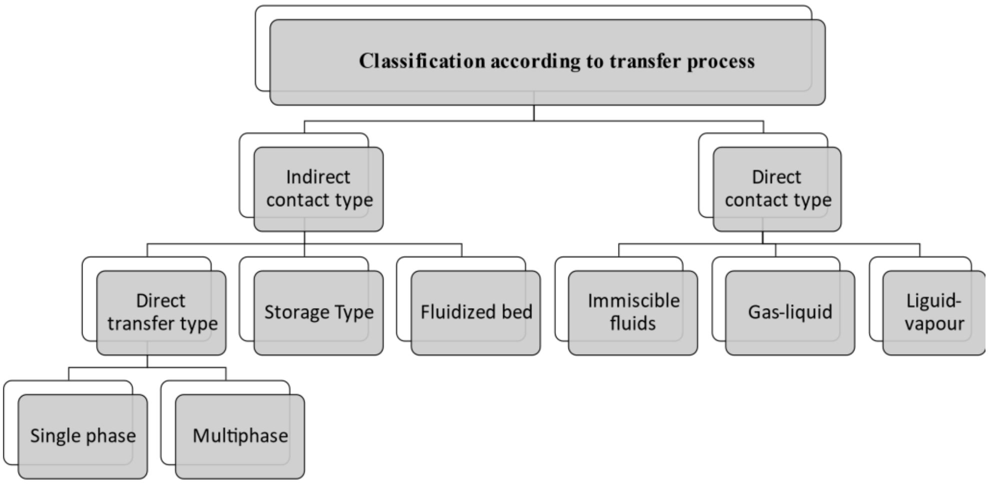

Figure 1. Classification according to transfer process.

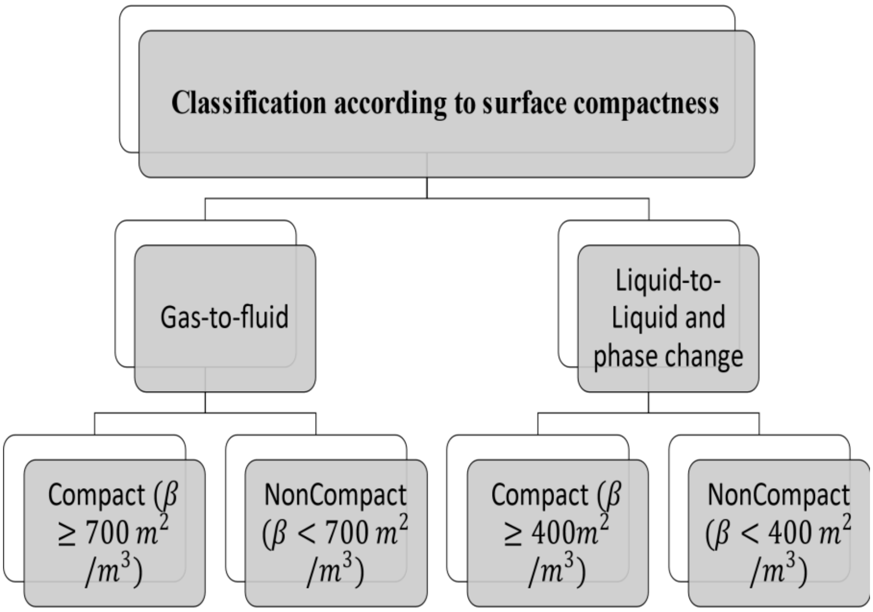

Figure 2. Classification according to surface compactness.

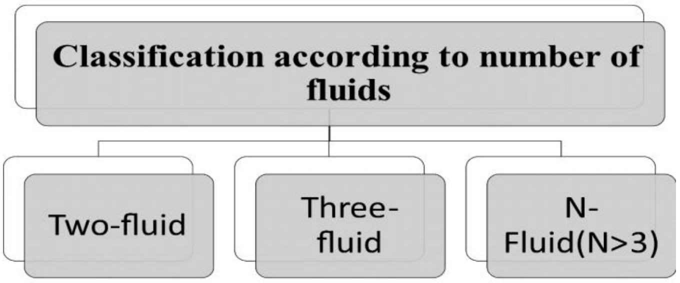

Figure 3. Classification according to number of fluids.

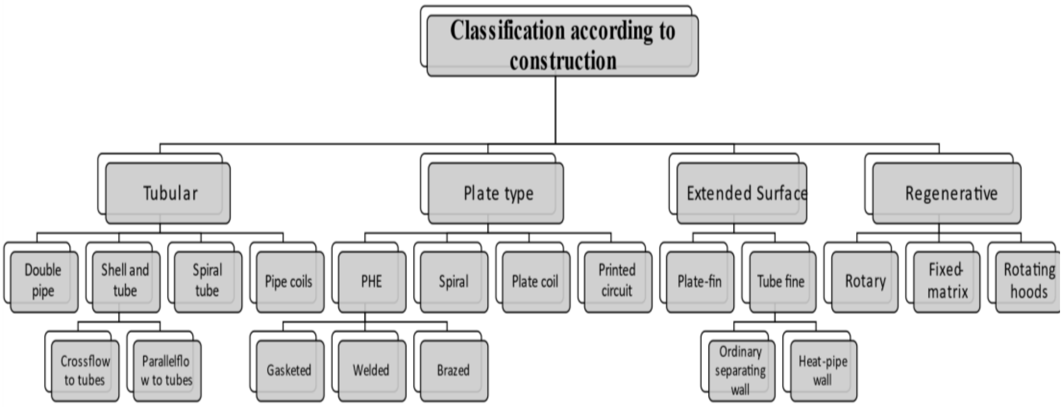

Figure 4. Classifications according to construction.

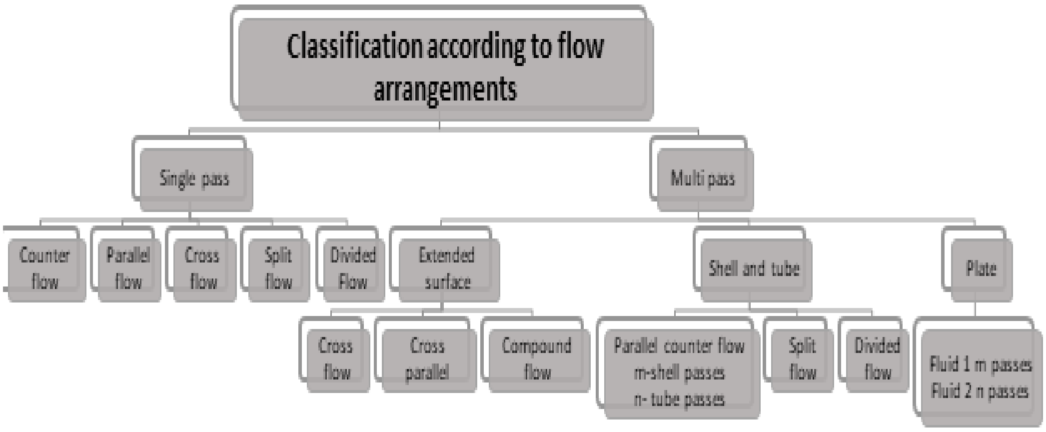

Figure 5. Classifications according to flow arrangements

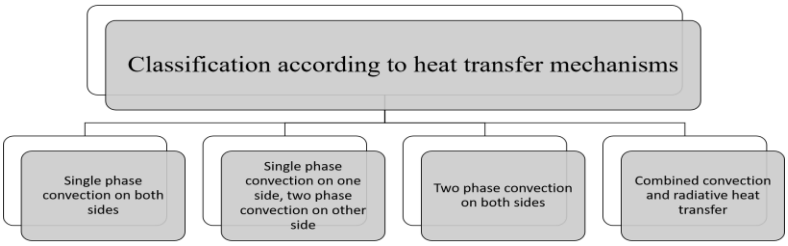

Figure 6. Classifications according to heat transfer mechanisms.

The shell and tube exchanger consists of four major parts:

1) Front Header—this is where the fluid enters the tube side of the exchanger. It is sometimes referred to as the Stationary Header.
2) 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.
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. Tubular Exchanger Manufacturers Association (TEMA) letter designation:

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.

Shell-and-tube exchangers are classified and constructed in accordance with the widely used TEMA (Tubular Exchanger Manufacturers Association) standards. In all three types, the front-end head is stationary while the rear-end head can be either stationary or floating, depending on the thermal stresses in the shell, tube, or tube sheet, due to temperature differences as a result of heat transfer.

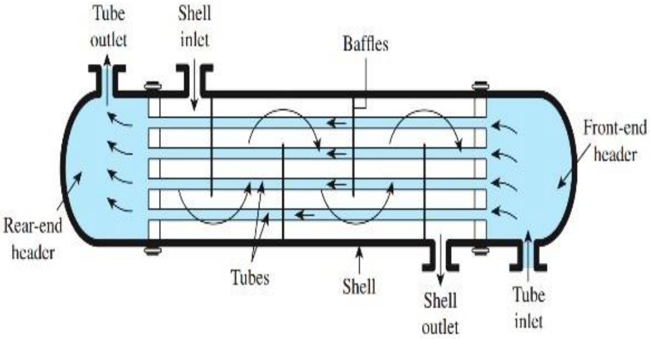

Figure 7. Schematic of Shell and Tube heat exchanger.

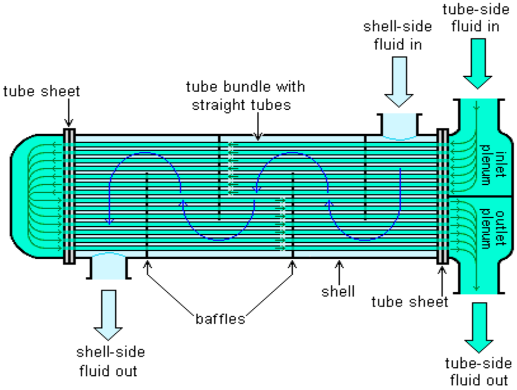

Figure 8 Straight tube heat exchanger (two pass tube side)

The three most common types of shell-and-tube exchangers are (1) fixed tube sheet design, (2) U-tube design, and (3) floating-head type.

B. Applications of heat exchangers

Shell and tube heat exchangers are considered one among the most effective type of heat exchangers. These heat exchanges have a cylindrical shell with a bundle of tubes. The tubes are made from thermally conductive materials, which allow heat exchange between the hot fluids flowing outside the tubes and the coolant flowing through the tubes [4].

These heat exchangers offer an optimal cooling solution to different applications including:

- Hydraulic
- Leisure
- Marine
- Rail
- Industrial

Shell and tube heat exchangers are widely used in applications, which require cooling, or heating large volume of process fluids or gases. There are a number of different types of shell and tube heat exchanger designs to meet several processes needs in virtually every industry.

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.[5]

Shell and tube heat exchanger is used in various industrial process applications because they can perform tasks such as:

- Removal of process heat and feed water preheating
- Cooling of hydraulic and lube oil
- Cooling of turbine, compressor, and engine
- Condensing process vapor or steam

Evaporating process liquid or steam 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.

II. EXPERIMENTAL METHODOLOGY DESCRIPTION

Heat exchangers are devices that transfer energy between fluids at different temperatures by heat transfer. Heat exchangers may be classified according to different criteria. The classification separates heat exchangers in recuperators and regenerators, according to construction is being used. In recuperators, heat is transferred directly (immediately) between the two fluids and by opposition, in the regenerators there is no immediate heat exchange between the fluids.

Rather this is done through an intermediate step involving thermal energy storage. Recuperators can be classified according to transfer process in direct contact and indirect contact types. In indirect contact heat exchanger, there is a wall (physical separation) between the fluids. The recuperators are referred to as a direct transfer type. [6]

In contrast, the regenerators are devices in which there is intermittent heat exchange between the hot and cold fluids through thermal energy storage and release through the heat exchanger surface or matrix. Regenerators are basically classified into rotary and fixed matrix models. The regenerators are referred to as an indirect transfer type.

The basic design methods for two fluid heat exchangers are:

1) log-mean temperature difference (LMTD) method
2) the effectiveness $\varepsilon-$NTU method
3) dimensionless mean temperature difference ($\Psi-$P)
4) (P1 – P2) to analyze recuperations.

Heat exchangers are usually analyzed using either the Logarithmic Mean Temperature Difference (LMTD) or the Effectiveness – Number of Transfer Units (ε-NTU) methods. The LMTD method is convenient for determining the overall heat transfer coefficient based on the measured inlet and outlet fluid temperatures. The ε-NTU method is more convenient for prediction of the outlet fluid temperatures if the heat transfer coefficient and the inlet temperatures are known.[7]

The analysis presented below assumes that:

1. There is no energy loss to the environment

2. Heat exchanger is at a steady-state
3. There are no phase changes in the fluids
4. Heat capacities of the fluids are independent of temperature
5. Overall heat transfer coefficient is independent of the fluid temperature and position within the heat exchanger.

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.[8]

A. *Governing equations*

The energy rate balance is:

$\frac{dE_{cv}}{dt} = Q - W + \sum_i m_i \left(h_i + \frac{v_i^2}{2} + gz_i\right) - \sum_e m_e \left(h_e + \frac{v_e^2}{2} + gz_e\right)$ ---------(1)

For a control volume at steady state, $\frac{dE_{cv}}{dt}$=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˙=0 and single-stream (only one inlet and one exit) and from the steady-state form the heat transfer rate becomes simply [9]:

$\dot{Q} = \dot{m}(h_2 - h_1)$-----(2)

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

For hot fluids,

$\dot{Q} = \dot{m}(h_{h1} - h_{h2})$ ---------(3)

For cold fluids,

$\dot{Q} = \dot{m}(h_{c2} - h_{c1})$ --------(4)

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

$\dot{Q} = UA\Delta T_{lm}$ ----------(5)

where U is the overall heat transfer coefficient, whose unit is w/m^2 °C and is ΔT_{lm} log-mean temperature difference.

B. *Overall heat transfer coefficients*

A heat exchanger involves two flowing fluids separated by a solid wall. Heat is transferred from the hot fluid to the wall by convection, through the wall by conduction and from the wall to the cold fluid by convection.

$UA = U_oA_o = U_iA_i = \frac{1}{R_t}$ -----------(6)

where $A_i = \pi D_i L$ and $A_o = \pi D_o L$and U is the overall heat transfer coefficient based on that area. $R_t$is the total thermal resistance and can be expressed as [1]

$R_t = \frac{1}{UA} = \frac{1}{h_iA_i} + R_w + \frac{R_{fi}}{A_i} + \frac{R_{fo}}{A_o} + \frac{1}{h_oA_o}$ ----------(7)

where R_f is fouling resistance (factor) and R_w is wall resistance and is obtained from the following equations.

For a bare plane wall.

$R_w = \frac{t}{KA}$ -------------(8)

where t is the thickness of the wall

For a cylindrical wall

$R_w = \frac{\ln(\frac{r_o}{r_i})}{2\pi LK}$ ------------(9)

The overall heat transfer coefficient based on the outside surface area of the wall for the unfinned tubular heat exchangers,

$U_o = \frac{1}{\frac{r_o}{r_i}\frac{1}{h_i} + \frac{r_o}{r_i}R_{fi} + \frac{r_o}{k}\ln\left(\frac{r_o}{r_i}\right) + R_{fo} + \frac{1}{h_o}}$ -------(10)

where R_{fi} and R_{fo} are fouling resistance of the inside and outside surfaces, respectively.

or

$U_o = \frac{1}{\frac{r_o}{r_i}\frac{1}{h_i} + R_{ft} + \frac{r_o}{k}\ln\left(\frac{r_o}{r_i}\right) + \frac{1}{h_o}}$ ---------(11)

where R_{ft} is the total fouling resistance, given as

$R_{ft} = \frac{A_o}{A_i}R_{fi} + R_{fo}$ -------(12)

For finned surfaces,

$\dot{Q} = \eta hA\Delta T$ ------------(13)

where η is the overall surface efficiency and

$\eta_f = \frac{\dot{Q}_f}{\dot{Q}_{fmax}}$ --------(14)

$\eta = 1 - \frac{A_f}{A}(1 - \eta_f)$ ----------(15)

Constant cross-section of very long fins and fins with insulated tips, the fin efficiency can be expressed as

$\eta_{f,insulated} = \frac{\tanh(mL)}{mL}$ -----(16)

$\eta_{f,long} = \frac{1}{mL}$ ------(17)

where L is the fin length.

For circular fins of rectangular profile,

$\eta_{f,rectangular} = C\frac{K_1(mr_1)I_1(mr_{2c}) - I_1(mr_1)K_1(mr_{2c})}{I_0(mr_1)K_1(mr_{2c}) - K_0(mr_1)I_1(mr_{2c})}$ --------(18)

For straight parabolic fins,

$\eta_{f,Parabolic} = \frac{2}{1+\sqrt{(2mL)^2+1}}$ ---------(19)

For straight triangular fins,

$\eta_{f,triangular} = \frac{1}{mL}\frac{I_1(2mL)}{I_0(2mL)}$ --------(20)

where the mathematical functions I and K are the modified Bessel functions and

$m = \sqrt{2h/kt}$ ---------(21)

where t is the fin thickness.

and

$r_{2c} = r_2 + t/2$ --------(22)

So,

$C = \frac{2r_1/m}{{r_{2c}}^2 - {r_1}^2}$ -----------(23)

For pin fins of rectangular profile,

where

$m = \sqrt{4h/kD}$ --------(24)

So,

$$\eta_{f,pin,recctangular} = \frac{\tanh mL_c}{mL_c} \text{------(25)}$$

and corrected fin length, Lc, defined as

$$L_c = L + D/4 \text{ -----------(26)}$$

The corrected fin length is an approximate, yet practical and accurate way of accounting for the loss from the fin tip is to replace the fin length L in the relation for the insulated tip case.

A is the total surface area on one side

$$A = A_u + A_f \text{--------------(27)}$$

The overall heat transfer coefficient is based on the outside surface area of the wall for the finned tubular heat exchangers,

$$U_o = \frac{1}{\frac{A_o}{A_i}\frac{1}{\eta_i h_i} + \frac{A_o R_{fi}}{A_i\ \eta_i} + A_o R_w + \frac{R_{fo}}{\eta_o} + \frac{1}{\eta_o h_o}} \text{ ---------(28)}$$

III. COMPUTATIONAL FLUID DYNAMICS (CFD) APPLICATION IN HEAT EXCHANGERS STUDY:

Due to difficulty involving fluid flow, heat transfer and associated phenomena such as chemical reactions occurred in experimental study of Heat exchangers. Means of computer-based simulation (CFD) is considered as a very powerful techniques span wide range of industrial and non-industrial application areas. CFD analysis is best at solving familiar heat exchanger problems. Heat Transfer Research, Inc uses CFD simulation effectively used in research and contract projects to improve heat exchanger performance [10].

Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.). Computational. It's the finite element model technique which is great promise in the study of the fluids flow, heat transfer and etc. Generally finite element model describes the model of any body by dividing the body into small parts known as elements. Meshing is the discretization of the domain into small volumes where the equations are solved. CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: (i) a pre-processor, (ii) a solver and (iii) a post-processor. We briefly examine the function of each of these elements within the context of a CFD code.[11]

There are three types of grids: structured grids, unstructured grids and block structured grids. The simplest one is structured grid. This type of grids, all nodes have the same number of elements around it. We can describe and store them easily. But this type of grid is only for the simple domain. If we have a complex domain, we can use unstructured grid. Generally, unstructured grid is suitable for all geometries. It is very popular in CFD. The disadvantage is that because the data structure is irregular, it is more difficult to describe and store them. Block structure grid is a compromising of structured and unstructured grid [12].

The idea is, firstly, divide the domain into several blocks, then use different structured grids in different blocks. Modeling starts with describing the boundary and initial conditions for the dominion and leads to modeling of the entire system. The typical boundary conditions in CFD are No-slip boundary condition, Axisymmetric boundary condition, Inlet, outlet boundary condition and Periodic boundary condition. 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 described by an average flow direction [13].

A. Using CFD tools for studying Heat Exchanger Design Modification for Performance Optimization [14]

In this study, it's concentrated on study of performance of shell and tube heat exchanger using CFD tool. The practical data as well as the design are taken from previous studies which are taken from reference paper. The application of CFD tools is justifying in the design of heat exchanger system with different conditions. First, the study and analysis of the heat exchangers, then selection of modeling of heat exchanger system as per reference paper. Second will contained with CFD analysis of the systems. Then comparison between experimental and CFD tools to show the effective application of the CFD tools. The problem is known as per physical dimensions in industry and conduct CFD analysis as per available data. Also, the analysis should also be conducted for different operating conditions like parallel flow and counter flows and varied baffle geometric conditions.

The industry required to modify their heat exchanger using CFD tools and compare results to develop heat exchanger system. So, it was taken as problem statement as modelled and analyzed under the actual boundary conditions to evaluate CFD results. The results of CFD and actual results has to be synchronized.

Furthermore, the operating conditions of the systems was varied and calculated to figure out the best solution. The modification in the baffle design with spherical dimples were the better performance solution compared to others.

Also, the rectangular cross section for the tube in designing was a better performance than all other cases. Hence, it can be concluded that, the rectangular tube is having better performance than the circular tube and the modifications in the baffle designs can be implemented to get the better performance.

B. Studying Performance of Shell and Tube Heat Exchangers with Varying Tube Layouts Using CFD [15]

In this study, heat exchanger is vertically cut-plane sections for velocity distributions of tube side fluid. Shell and tube heat exchangers increase in fluid velocity which result in decrease in fluid density because shell side is heated up. active zones, portions with high velocity were noticed that it occurs between the baffles because of heating of cross flow.

The pressure distribution of the vertical cut plane section of heat exchanger shell. It's noticed that pressure drop is obvious in shell zones than in baffle window because of tube bundles obstruct the flow. Also, it's observed that the region where Reynolds number is less than 1000 the performance factors of shell and tube heat exchanger rotated triangular and shell and tube heat exchanger combined decrease sharply. However, in high Reynolds number it's decrease slowly. Shell and tube heat exchanger triangular has higher performance than others.

Heat transfer coefficient increases at a faster rate in lower pressure drop region than the higher pressure drop region. This confirms some facts found in the literature that as the mass flow or Reynolds number increases, the increase in heat transfer rate decreases with the increase in pressure drop. Shell and tube heat exchanger triangular has a higher heat transfer coefficient than the other two heat exchangers under the same pressure drop. However, shell and tube heat exchanger combined is also higher than shell and tube heat exchanger rotated triangular under the same pressure drop

Numerical studies have been conducted to predict the performance of shell-and-tube heat exchangers with three different tube layout modes. The results show that most of the heat transfer and pressure drop occur during the cross flow of the shell fluid through the tube bundle. Compared with other shell and tube heat exchangers triangular, the average deviations of the heat transfer coefficients of shell and tube heat exchangers rotated triangular and shell and tube heat exchangers combined are 11.2% and 8.3%, respectively, while the pressures of shell and tube heat exchangers rotated triangular and shell and tube heat exchangers combined have dropped by 16.0% and 18.8%, respectively.

Evaluate performance from two selected criteria in heat exchangers, shell and tube heat exchangers triangular is more ideal, followed by shell and tube heat exchangers combined, because for the same shell side pressure drop, they show a higher heat transfer coefficient than shell and tube heat exchangers rotated triangular.

C. CFD analysis of a small shell and tube Heat exchanger with special shell side [16]

In this study, in order to enhance the model detail and make solid observations on the flow inside the shell, a compact heat exchanger is chosen. Some of the design parameters as well as the geometric parameters that have been selected. Six baffles in what seems like a geometric model. There are two alternative baffle cut values to choose from: In shell-and-tube heat exchanger designs, a 25% baffle cut value is popular; however, a 36 % The baffle cut value is set to either below or slightly above the central row of tubes. Water is the operating fluid on the shell side. Because the properties of water in the Fluent database are defined as constants, they are redefined using piecewise-linear functions of temperature in the "Thermo-Physical Properties of Saturated Water" tables to increase accuracy.

The accuracy of a comprehensive CFD model for a laminar flow heat exchanger was verified by comparing the results with actual experimental data from a very small educational demonstration unit with two baffles for verification of our general CFD modelling approach. The sensitivity of the results to the turbulence model and the discretization order for three distinct shell side mass flow rates is explored as the first step of the current study for a larger heat exchanger with turbulent flow. Variations of the shell side heat transfer coefficient and pressure drop values with flow rate and baffle spacing are then calculated using the chosen turbulence model and discretization scheme. Finally, the effects of baffle cut on the heat transfer and the pressure drop was selected.

The results' sensitivity to the turbulence model and discretization orders the flow inside the shell seems to be turbulent in all preliminary simulations. As a result, the turbulent viscous model is chosen. The heat exchanger model with six baffles is used to evaluate the sensitivity of the results to the turbulence model and the discretization order. For two different mesh densities, the first and second order discretization schemes, as well as three distinct turbulence models, are tested.

A small shell-and-tube heat exchanger's shell side is modelled sufficient accuracy to resolve the flow and temperature fields. Shell side heat transfer coefficient, pressure drop, and heat transfer rate data are computed using CFD simulation results for fixed tube wall and shell inlet temperatures.

The mesh density, order of discretization, and turbulence modelling all affect the sensitivity of the shell side flow and temperature distributions. For the first and second order discretization, three turbulence models are tested using two different mesh densities. The k-" realizable turbulence model with first order discretization and fine mesh is selected as the best simulation approach. The results of the simulation are compared with those of the Kern and Bell-Delaware methods. It has been discovered that the Kern method consistently underestimates the heat transfer coefficient.

The CFD simulation results for suitably spaced baffles are shown to be in extremely good agreement with the Bell-Delaware results. The baffle cut selection affects the results; for this heat exchanger design, a 25% baffle cut provides slightly better results. The majority of the time, the variations between Bell-Delaware and CFD estimates of the overall heat transfer rate are less than 2%.

The flow structures shown using CFD simulations revealed that the cross-flow windows are not efficiently utilized for the reduced number of baffles, and certain recirculation zones form behind the baffles. This flaw is rectified by increasing the number of baffles, and the heat transfer characteristics of the heat exchanger are enhanced. In general, correlation-based techniques can indicate the presence of a design flaw, but CFD simulations can pinpoint the cause and position of the flaw.

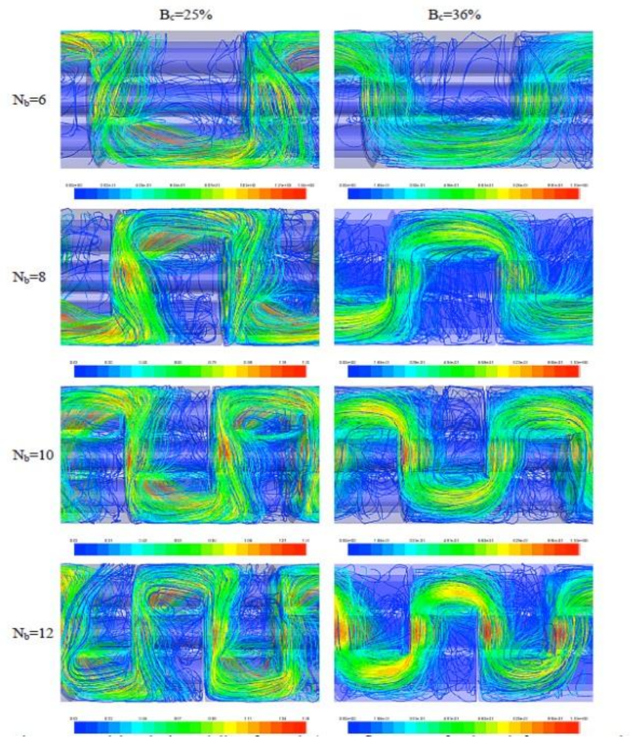

Figure 9: Particle velocity path lines for 0.5 kg/s mass flow rate. Left column is for B_c=25% and right column is for B_c=36%. Rows from top to bottom are for Nb=6,8,10 and 12.

D. *Temperature Analysis of Shell and Tube Heat Exchanger [17]:*

In this study, a shell and tube heat exchanger installed at a particular pipe manufacturing company has been considered. The industry involves a variety of different manufacturing processes. Oil is used to lubricate the bearings in the compressors used in the VCM (Vinyl Chloride Monomer) manufacturing plant.

A shell and tube heat exchanger is used to lower the temperature of the oil using water as the temperature of the oil increases. This cooled oil is utilized to lubricate compressor bearings once more. The heated water is then sent to the cooling towers.

The heat exchanger was installed when the plant launched in 1994, and no changes have been made to it since then. The goal of this research is to investigate the heat exchanger and determine its overall heat transfer coefficient based on current operating conditions. A great deal of research has been done on heat exchangers and their optimization in terms of thermal performance overview of such devices.

Based on the results of the heat exchanger flow simulation, it can be determined that the simulation produces results that are similar to those achieved through experimental.

Furthermore, different parameters such as baffle spacing and tube pitch layout have an impact on the overall heat transfer coefficient. It has been discovered that while utilizing square pitch, the overall heat transfer coefficient reduces, whereas it increases when the baffle spacing lowers.

E. *Performance Analysis of Shell & Tube Type Heat Exchanger under the Effect of Varied Operating Conditions [18]*

In this study, concerning the performance analysis of shell and tube type heat exchangers under various loading circumstances. To accomplish so, first built a shell and tube type heat exchanger to determine the dimensions of the parts involved, and then fabricated and tested the actual working model to evaluate how various parameters affected the heat exchanger's performance.

A heat exchanger's thermal performance is influenced by a variety of factors. Thermal conductivities of included fluids and materials, velocity of flow, turbulence, quality and quantity of provided insulation, ambient circumstances, flow conditions, and so on are some of them. It's seldom easy to make an exact estimate about a heat exchanger's performance under a set of loading conditions. However, certain testing and experience can be used to make predictions up to a certain level. The purpose of this research is to examine the performance of a shell and tube type heat exchanger under specific loading conditions.

It is obvious that many factors influence the heat exchanger's performance, and the formulas' effectiveness represents the cumulative effect of all factors on the heat exchanger's performance. Insulation is a good tool for increasing the rate of heat transmission if utilised properly and much below the critical thickness limit. Cotton wool and tape have provided the highest levels of effectiveness among the materials used. Furthermore, the heat exchanger's efficiency is determined by the amount of turbulence present. However, it is clear that there is no direct relation between turbulence and effectiveness, and that effectiveness peaks at a intermediate value. The ambient conditions in which the heat exchanger was tested had no apparent effect on the heat exchanger's performance.

F. *Shell & tube heat exchanger thermal design with optimization of mass flow rate and baffle spacing[19]*

In this study, according to earlier studies, increasing the fluid flow rate in a shell and tube heat exchanger causes an increase in pressure drop, which increases pumping power. When compared to traditional designs, the genetic algorithm produces significantly better optimal designs. The use of a genetic algorithm to determine the global minimum heat exchanger cost is much faster and has an advantage over other methods in terms of finding several solutions of the same quality. As a result, the designer has additional options.

In most circumstances, determining NTU requires extensive computations that might be time consuming. The developed charts and tables assist engineers save time by allowing them to choose the needed NTU and capacity rate ratio without having to perform complex calculations. Once these two values are determined, the designer has a starting point from which to extrapolate the heat exchanger's detailed design. Similarly, technical engineers must understand the heat exchanger equipment so that it can be operated and maintained within its performance range. The throughput of any process sector must be changed on a regular basis based on demand. The heat exchanger will be forced to vary from its original design state as a result of this. Engineers can use performance charts and tables to help them in these circumstances

G. Shell Design of Shell and Tube Type Heat Exchanger using CFD Tools [20]:

In this study, Correlations are extensively used in the design of shell and tube heat exchangers, with the Kern and Bell-Delaware methods being the most popular. The Kern method is typically used for preliminary design and provides conservative results. The Bell-Delaware method, on the other hand, is a more accurate method which can provide precise results. It can more accurately predict and estimate pressure drop and heat transfer coefficient. The Bell-Delaware method is a rating method that can suggest weaknesses in the shell side design but cannot specify where these weaknesses are located. Flow distribution must therefore be studied in order to solve these issues. As a result, a number of analytical, experimental, and numerical studies have been conducted.

The majority of this research focused on specific aspects of shell and tube heat exchanger design. In general, these correlations are created for baffled shell and tube heat exchangers. Heat transfers and flow distribution are thoroughly discussed, and the proposed model is compared to increasing baffle inclination angle. With an average inaccuracy of 20%, the model accurately predicts heat transfer and pressure drop. As a result, there is room for improvement in the model. Except for the outlet and inlet regions, where rapid mixing and flow direction change occurs, the assumption worked well in this geometry and meshing. Thus, if the helical baffle used in the model has entire contact with the surface of the shell, more turbulence across the shell side will be generated, and the heat transfer rate will be increased.

If using a different flow rate, the results will be different, it could assist in increasing heat transfer and the temperature difference between the input and output. Because most of the fluid passes through the baffles without interacting with them, the heat transfer rate is poor. As a consequence, the design can be adjusted for better heat transmission in two ways: by reducing the shell diameter to ensure proper contact with the helical baffle, or by increasing the baffle to ensure proper contact with the shell. It's because the heat transfer area isn't being used to its full potential. As a result, cross-flow regions can be added to the design to ensure that flow does not remain parallel to the tubes. It will allow the outer shell fluid to get in contact with the inner shell fluid, leads to faster heat transfer rate.

IV. CONCLUSIONS

According to previous studies presented in the following research. It was found that, CFD is a powerful tool can be used in prediction of characteristic and performance for a Heat exchanger. Thus, heat transfer coefficient and pressure drop both of them increases with higher mass flow rate. While, increasing heat transfer co-efficient is not a valuable solution as the system consumed power is also increasing with higher pressure drop. It was found also that, changes in baffle figures affect the shell side pressure drop strongly. So, we have to compare type of geometry which permits low pressure drop.

Moreover, The helical baffle type with high heat transfer co-efficient than segmental baffle because of its more turbulence in shell side flow, due to high turbulence the fouling is lower compare to segmental baffle and due to less fouling the heat transfer co- efficient is higher, The advantage of reducing the pressure drop are like pumping efforts is reduced for pumping the shell side flow, so the overall efficiency of the heat exchanger can be increased, other reason is due to more turbulence in the shell side flow the fluid is passing towards all the surface of tube in short by passing through reduced and it result in higher heat transfer co-efficient.

NOMENCLATURE

- $\dot{Q}$= 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
- $\dot{m}$ = 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
  R_f= fouling resistance (factor)
- R_w = wall resistance
- t = the thickness of the wall
- R_{fi} and R_{fo} = fouling resistance of the inside and outside surfaces
- R_{ft}=the total fouling resistance
- A_f=fin surface area
- η_f= fin efficiency
- L= fin length
- Lc= corrected fin length
- D = the diameter of the cylindrical fins
- A_o and A_i = the total surface area of the outer and inner surfaces

REFERENCES

[1] R. K. Shah, and D. P. Sekulic, *Fundamentals of heat exchanger design*: John Wiley & Sons, 2003.

[2] A. L. Costa, and E. M. Queiroz, “Design optimization of shell-and-tube heat exchangers,” *Applied thermal engineering,* vol. 28, no. 14-15, pp. 1798-1805, 2008.

[3] F. Incropera, D. DeWitt, T. Bergman, and A. Lavine, *Fundamentals of Heat and Mass Transfer*, 2007.

[4] O. Khayal, *Fundamentals of Heat Exchangers*, 2018.

[5] K. Arjun, and K. Gopu, “Design of shell and tube heat exchanger using computational fluid dynamics tools,” *Res. J. Eng. Sci*, 2014.

[6] Y. Lei, Y. Li, S. Jing, C. Song, Y. Lyu, and F. Wang, “Design and performance analysis of the novel shell-and-tube heat exchangers with louver baffles,” *Applied Thermal Engineering,* vol. 125, pp. 870-879, 2017.

[7] P. Gupta, A. K. Sharma, and R. Kumar, “Shell Side CFD Analysis of a Small Shell-and-Tube Heat Exchanger with Elliptical Tubes,” International Journal of Performability Engineering, vol. 15, no. 9, pp. 2294, 2019.

[8] S. Rajasekaran, and T. Kannadasan, “A simplified predictive control for a shell and tube heat exchanger,” International Journal of Engineering Science and Technology, vol. 2, no. 12, pp. 7245-7251, 2010.

[9] U. U. Rehman, “Heat transfer optimization of shell-and-tube heat exchanger through CFD studies,” 2012.

[10] M. M. Aslam Bhutta, N. Hayat, M. H. Bashir, A. R. Khan, K. N. Ahmad, and S. Khan, “CFD applications in various heat exchangers

design: A review," Applied Thermal Engineering, vol. 32, pp. 1-12, 2012/01/01/, 2012.

[11] C. Abeykoon, "Compact heat exchangers–Design and optimization with CFD," International Journal of Heat and Mass Transfer, vol. 146, pp. 118766, 2020.

[12] S. Marzouk, M. Al-Sood, M. El-Fakharany, and E. El-Said, "A comparative numerical study of shell and multi-tube heat exchanger performance with different baffles configurations," International Journal of Thermal Sciences, vol. 179, pp. 107655, 09/01, 2022.

[13] M. B. Slimene, S. Poncet, J. Bessrour, and F. Kallel, "Numerical investigation of the flow dynamics and heat transfer in a rectangular shell-and-tube heat exchanger," Case Studies in Thermal Engineering, vol. 32, pp. 101873, 2022.

[14] M. E. Srinivasan S, Seenivasaperumal V, M.E2, Manikandan G, M.E3, "Heat Exchanger Design Modification for Performance Optimization Using CFD Tools," International Journal of Applied Engineering Research, vol. 13, 2018.

[15] M. Petinrin, and A. Dare, "Performance of Shell and Tube Heat Exchangers with Varying Tube Layouts," British Journal of Applied Science & Technology, vol. 12, pp. 1-8, 01/10, 2016.

[16] E. Ozden, and I. Tari, "Shell side CFD analysis of a small shell-and-tube heat exchanger," Energy Conversion and Management, vol. 51, no. 5, pp. 1004-1014, 2010.

[17] S. L. Chirag Raut , Tushar Dhande, "Study and Design of Shell and Tube Heat Exchanger," International Research Journal of Engineering and Technology (IRJET), vol. Volume: 08, no. 06 | June 2021, 2021.

[18] V. V. P. Dubey, R. R. Verma, P. S. Verma, and A. Srivastava, "Performance analysis of shell & tube type heat exchanger under the effect of varied operating conditions," IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), vol. 11, no. 3, pp. 08-17, 2014.

[19] P. A. M. M. Sandeep K. Patel, "SHELL & TUBE HEAT EXCHANGER THERMAL DESIGN WITH OPTIMIZATION OF MASS FLOW RATE AND BAFFLE SPACING " International Journal of Advanced Engineering Research and Studies, vol. II, no. I, 2012.

[20] A. S. Devvrat Verma, "Design of Shell and Tube Type Heat Exchangerusing CFD Tools," IJIRST– International Journal for Innovative Research in Science & Technology, vol. 4, no. 3, 2017.