Review Paper on Analysis of Heat Transfer in Shell and Tube Type Heat Exchangers

Durgesh J Bhatt[1] Prof. Priyanka M Jhavar[2] Prakash J Sakariya[3]
[1]PG Student [2]Professor [3]PG student
[1]Department of Mechanical Engineering
[1]Shree SatyaSaya Institute of Science and Technology, Sehore, India

Abstract— A heat exchanger is a device that is used to transfer thermal energy (enthalpy) between two or more fluids, at different temperatures and in thermal contact The tube diameter, tube length, shell types etc. are all standardized and are available only in certain sizes and geometry. And so the design of a shell-and-tube heat exchanger usually involves a trial and error procedure where for a certain combination of the design variables the heat transfer area is calculated and then another combination is tried to check if there is any possibility of reducing the heat transfer area. A primary objective in the Heat Exchanger Design (HED) is the estimation of the minimum heat transfer area required for a given heat duty, as it governs the overall cost of the HE. But there is no concrete objective function that can be expressed explicitly as a function of the design variables and in fact many numbers of discrete combinations of the design variables are possible as is elaborated below. Traditional optimization techniques do not ensure global optimum and also have limited applications. In the recent past, some experts studied on the design, performance analysis and simulation studies on heat exchangers. Modeling is a representation of physical or chemical process by a set of mathematical relationships that adequately describe the significant process behavior. These models are often used for Process design, Safety system analysis and Process control. A steady state model for the outlet temperature of both the cold and hot fluid of a shell and tube heat exchanger will be developed and simulated, which will be verified with the experiments conducted. Based on these observations correlations to find film heat transfer coefficients will be developed during any process of refining of chemical manufacturing. Then these models are simulated on computer software. In this problem of heat transfer involved the condition where different constructional parameters are changed for getting the higher heat transfer rate within the thermal and hydraulic stability. And various models developed according to the change in physical parameters & results are obtained. These models were developed using latest computers tools like ANSYS, Fluent, and MATLAB etc.. The obtained results were also evaluated by comparing the same with the industrial operating exchanger and found satisfactory.

Key words: Shell and tube heat exchanger, Performance analysis, ANSYS, CFD analysis

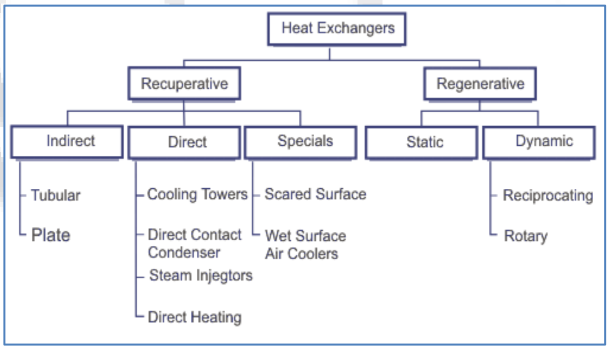
I. INTRODUCTION

A. Heat exchanger

Transfer of heat from one fluid to another is an important operation for most of the chemical industries. The most common application of heat transfer is in designing of heat transfer equipment for exchanging heat from one fluid to another fluid. Such devices for efficient transfer of heat are generally called Heat Exchanger.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 Recuperator. 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.

B. Types of Heat Exchanger

- According to construction features
- According to heat transfer mechanisms
- According to flow arrangements
- According to transfer processes
- According to surface compactness
- According to number of fluids



Tabular exchangers are generally built of circular tubes, although elliptical, rectangular, orround/flat twisted tubes have also been used in some applications. There is considerableflexibility in the design because the core geometry can be varied easily by changing thetube diameter, length, and arrangement. These exchangers may be classified as shell-andtube,double-pipe, and spiral tube exchangers. They are all prime surface exchangersexcept for exchangers having fins outside/inside tubes.

C. Construction

This exchanger, shown in Fig. 1.1, is generallybuilt of a bundle of round tubes mounted in a cylindrical shell with the tube axis parallelto that of the shell. One fluid flows inside the tubes, the other flows across and along thetubes. The major components of this exchanger are tubes (or tube bundle), shell, frontendhead, rear-end head, baffles, and tube sheet. 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 methodsemployed to reduce thermal stresses, to prevent leakages, to provide for ease of cleaning,to contain operating pressures and temperatures, to control corrosion, to accommodatehighly asymmetric flows, and so on.

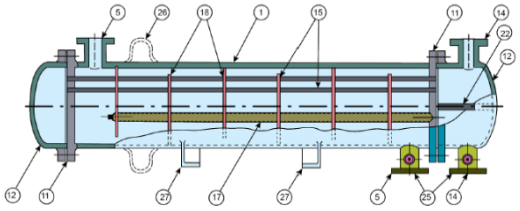


Fig. 1.1: Shell and Tube heat Exchanger construction

Shell-and-tube exchangers are classified and constructedin accordance with the widely used TEMA (Tubular Exchanger ManufacturersAssociation). TEMA has developed a notation system to designate major types ofshell-and-tube exchangers. In this system, each exchanger is designated by a three-lettercombination, the first letter indicating the front-end head type, the second the shelltype, and the third the rear-end head type. Some commonshell-and-tube exchangers are AES, BEM, AEP, CFU, AKT, and AJW.

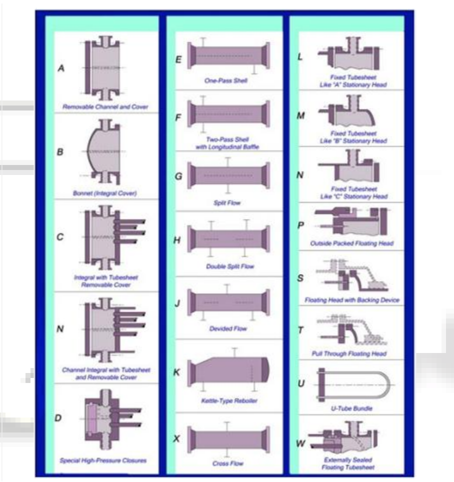


Fig. 1.2 : TEMA designation of exchanger

D. Operating Range

Generally the shell and heat type heat exchanger are widely used for various purposes having limitation to be designed for maximum up to 15000 psi and 1000 ^{0}F. Beyond above given parameter special consideration is required for the design of heat exchanger. Along with the same there are few disadvantages concerned with shell and tube heat exchangers are:

Fixed tube sheet type exchanger having cleaning issues as well as can be used for low temperature difference as there is no provision for the thermal expansion of the same.

Shell and tube heat exchanger are Subject to flow induced vibration which can lead to equipment failure.Subject to flow mal-distribution especially with two phase inlet streams

E. Application

– The design is ideal for high pressure and temperature services.
– Shell and tube heat exchanger are easy to clean for floating head type configuration so, can be used in dirty services.
– Shell and tube type heat exchanger can be used for higher temperature difference services as it can accommodate thermal expansion.
– They are most suitable for gas services and phase change service.

They can be designed for special operating conditions: vibration, heavy fouling, highly viscous fluids, erosion, corrosion, toxicity, radioactivity, multicomponent mixtures, and so on. They are used extensively as process heat exchangers in the petroleum-refining and chemical industries; as steam generators, condensers, boiler feed water heaters and oil coolers in power plants; as condensers and evaporators in some air-conditioning and refrigeration applications; in waste heat recovery applications with heat recovery from liquids and condensing fluids; and in environmental control.

II. Literature Review

A Shell and tube heat exchanger is a device in which energy is transferred from one fluid to another across a solid surface. Exchanger analysis and design therefore involve both convection and conduction. Two important problems in heat exchanger analysis are (1) rating existing heat exchangers and (ii) sizing heat exchangers for a particular application. Rating involves determination of the rate of heat transfer, the change in temperature of the two fluids and the pressure drop across the heat exchanger. Sizing involves selection of a specific heat exchanger from those currently available or determining the dimensions for the design of a new heat exchanger, given the required rate of heat transfer and allowable pressure drop. The LMTD method can be readily used when the inlet and outlet temperatures of both the hot and cold fluids are known. The choice of heat exchanger type directly affects the process performance and also influencesplant size, plant layout, length of pipe runs and the strength and size of supporting structures. These paper incorporate various design options for the heat exchangers including the variations in the tube diameter, tube pitch, shell type, number of tube passes, baffle spacing, baffle cut, etc.

A. Review Papers

Ebieto, C.E. and Eke G.B. [1] in his experimental paper the performance analysis carried out of shell and tube heat exchanger & analytical method was used to develop correlation for the performance analysis.

The thermal analysis of a shell and tube heat exchanger involves the determination of the overall heat-transfer coefficient from the individual film coefficients, and(Kern, 1965). The shell-side coefficient presents the greatest difficulty due to thievery complex nature of the flow in the shell. In addition, if the exchanger employs multiple tube passes, then the LMTD correction factor must be used in calculating the mean temperature difference in the exchanger. For the turbulent flow regime ($Re \geq 10^4$), the following correlation is widely used (serth, 2007).

$Nu = Re^{0.8}\, Pr^{1/3}\, (\mu/\mu_w)^{0.14}$(1)

Where ,

Nu = Nusselt Number = hd/k

Re = Reynold’s Number = $DV\rho/\mu$

Pr = Prandtl Number = $c_p\mu/k$

D = Inside diameter of the pipe

V = average fluid velocity.

C_p,μ,ρ,k = Fluid properties at avg. bulk temperature.

μ_w = Fluid viscosity evaluated at average wall temperature.

Eq. (1) holds good for 0.5 ≤ Pr ≥ 17,000 & for pipes L/D ≥ 10. However for short pipes 10 ≤ L/D ≥ 60, the right hand side of the equation is often multiplied by the factor [$1+(D/L)^{2/3}$] to correct for the entrance and exit effects. (Serth 2007).

For laminar flow in circular pipes (Re < 2100), the seider-Tate correlation takes the form : Nu = $1.86[RePrD/L]^{1/3}$ $(\mu/\mu_w)^{0.14}$. This equation is valid for 0.5<Pr<17000 and

$[RePrD/L]^{1/3}$ $(\mu/\mu_w)^{0.14}$ (2)

For low in he transition region (2100 < Re < 10^4), the Hausen correlation is :

$Nu = 0.116(Re^{2/3} - 125)\ Pr^{1/3}(\mu/\mu_w)^{0.14}(1+(D/L)^{2/3})$(3)

In computing the tube-side coefficient *hi* it isnassumed that all tubes in the exchanger are exposed to the same thermal and hydraulic conditions. The value of *hi* is then the same for all tubes, and the calculation can be made for a single tube. Equations (1), (2), or (3) were used, depending on the flow regime. The tube fluid heat transfer coefficient, *hi*, can be calculated using;

$hi= (NuK/D_i)$ (4)

The Delaware method (Serth, 2007) was used to compute the shell-side heat transfer coefficient, *ho* . In the equation for the overall heat transfer coefficient, the temperature difference, Δtm , is the mean temperature difference between the two fluid streams. Since is independent of position along the exchanger, Δtm is the logarithmic mean temperature difference (Serth, 2007);

$$\Delta Tm = LMTD = \frac{(T1-t2)-(T2-t1)}{IN\left(\frac{T1-t2}{T2-t1}\right)}(5)$$

Equation (5) is valid regardless of whether counter flow or parallel flow is employed. In multi-pass shell and-tube exchangers, the flow pattern is a mixture of co-current and countercurrent flow. For this reason, the mean temperature difference is derived by introducing a correction factor, *F*, which is termed the LMTD correction factor;

$\Delta Tm = F * \Delta Tm$ (6)

The correction factor is a function of the shell and tube fluid temperatures, and the number of tube and shell passes. This is corrected using two dimensionless temperature ratios (Serth, 2007); let, *N = No. of shell side passes* then

$$S = \frac{t2-t1}{T1-t1}\ (7A)$$

$$R = \frac{T1-T2}{T2-t1}\ (7B)$$

For R ≠1

$$F = \frac{\sqrt{R^2+1}In\left(\frac{1-S}{1-RS}\right)}{R-1\left[\frac{2-S\left(R+1-\sqrt{(R^2+1)}\right)}{2-S\left(R+1-\sqrt{(R^2+1)}\right)}\right]}(8)$$

For R= 1

$$F = \frac{2\sqrt{2}}{(1-s)In\left[\frac{2-s(2-\sqrt{2})}{2-s(2-\sqrt{2})}\right]}(9)$$

The required overall heat transfer coefficient is given by ;

$$U_{req} = \frac{Q}{AF\Delta Tm}(10)$$

The clean overall heat transfer coefficient is given by ;

$$U_c = [\frac{Do}{hiDi} + \frac{Doin\left(\frac{Do}{Di}\right)}{2k} + \frac{1}{ho}]^{-1}(11)$$

And the design overall heat transfer coefficient is given by ;

$$U_D = \left(\frac{1}{U_c+R_D}\right)^{-1}(12)$$

The effect of fouling is allowed for in the design by including the inside and outside fouling coefficients. Kern (1965) presented typical values for the fouling factors for common process service fluids used in plane tubes (not finned tubes). The fouling factor for the exchanger is given as (Serth, 2007);

$$R_D = R_{Di}\left(\frac{Do}{Di}\right) + R_{D0}(13)$$

Design problems frequently include specifications of the maximum allowable pressure drops in the two streams. In that case, pressure drops for both streams would have to be calculated in order to determine the hydraulic suitability of the heat exchanger. The pressure drop due to fluid friction in the tubes is given by the length of the flow path set to the tube length times the number of tube passes (Serth, 2007).

$$\Delta_{pf} = \frac{fn_pLG^2}{2\rho D_i s\emptyset}(14)$$

Δ_{pf} = Pressure drop (Pa)

f = Darcy friction factor (dimensionless)

n_p = No. of tube passes (dimensionless)

L = Tube length (m)

G = Mass flux (kg/s m^2)

D_i = Tube inside diameter (m)

ρ = Density of water (kg/m^3)

s = Fluid specific gravity (dimensionless)

$\emptyset$ = Viscosity correction factor (dimensionless)

$\emptyset = (\mu/\mu_w)^{0.14}$ for turbulent and transition flow.

$\emptyset = (\mu/\mu_w)^{0.28}$ For laminar flow

For laminar flow friction factor is given by;

f = *64/Re*(16)

for turbulent flow (RE > 3000) following equation can be use ;

$f = 0.4137\ R_e^{-0.2385}$(17)

The minor losses on the tube side are estimated usingthe following equation:

$\Delta_{pf} = 0.5x10^{-4}a_rG^2/s$ (18)

wherea_r is the number of velocity heads allocated for minor losses.

Serth (2007) proposed the following expression forcomputing the shell-side pressure drop:

$\Delta_{pf} = \frac{f d_{op} G^2 (n_b+1)}{2\rho D_e s \emptyset}$(19)

The shell side friction factor formulae is given by ;

f = 144{$f_1 - 1.25\ (1 - B/De_e)(f_1 - f_2)$(20)

An approximate equation for $f_1 and\ f_2$ are as follows :

For $R_e \geq 1000$.

$$f_1 = (0.0076 + 0.000166 d_s) R_e{}^{-0.125}$$

$(8\ \leq d_e\ \leq 42)$ (21)

$$f_{12} = (0.0016 + 5.8\ x 10^{-5} 6 d_s) R_e{}^{-0.157}$$

$(8\ \leq d_e\ \leq 23.25)$(22)

For $R_e < 1000$.

f_1 = exp[0.092$(InR_e)^2$-1.48 InR_e-0.000526$d_e{}^2$ + 0.047$d_e - 0.0038$

$(8\ \leq d_e\ \leq 42)$ (23)

f_1 = exp[0.123$(InR_e)^2$-1.78 InR_e-0.00132$d_e{}^2$ + 0.0678$d_e - 1.34$

$(8\ \leq d_e\ \leq 42)$(24)

A computer program in MATLAB was developed to evaluate the performance of shell and tube heat

Exchangers. It is also to be taken in account that the finalized design should be thermally and hydraulically stable. The program was tested with field data from five industrial exchangers showed that the result obtained, compares reasonably with the actual performance data, thus, demonstrating that the program is reliable and can be applied in the performance analysis of shell and tube heat exchangers. In obtained results the clean and overall heat transfer coefficients are greater than the required overall coefficient. This implies that the heat exchangers are thermally suitable for the service they are being used for.

Shell and tube side pressure drops are lesser than the allowable pressure drop, the heat exchangers are hydraulically suitable for the service they are being used for.

M. Thirumarimurugan, T.Kannadasan and E.Ramasamy[2] an experimental study of convective heat transfer co-efficient for shell and tube type heat exchanger was carried out by authors.

Experiments were conducted on a 1-1Shell and Tube heat exchanger with different cold side flowrates and different compositions of fluid. The effect of these parameters on the shell outlet temperature, tube outlet temperature and overall heat transfer coefficients were studied. It was found thatcold fluid outlet temperature decreases and the overall heat transfer coefficient increases with increase in flow rate of cold fluid. Also the outlet temperature of cold fluid decreases and overall heat transfer coefficient increases with increase in composition of water. The overall effectiveness of heat exchanger was found to increase with decrease in composition of water. It was found that the Cross Flow Heat Exchanger is the most effective compared with the Shell and Tube Heat Exchanger. A mathematical model of this system is developed, simulated using MATLAB and compared with the experimental values. Finally a correlation for the calculation of film heat transfer coefficient is developed using dimensional analysis for tube side. The physical model equation was developed using dimensional analysis followed by leastsquare curve fitting experimental data as follows:

$Nu = 0.4232(Re)^{0.339}(Pr)^{0.3412}(x)^{0.003}$

In this paper they have selected modified spiral plate heat exchanger for experimental investigation. The schematic diagram and experimental set up parameters given below,

B. Results and discussion

The effect of different input variables on output variable are discussed in detail in the following sections. Heat exchanger effectiveness, the film coefficients for both hot and cold fluids and overall heat transfer coefficient .

1) Effect of flow rate of the cold fluid:

Increase in theflow rate of cold fluid results in increase in the overallheat transfer coefficient as can be seen from tables. Thisis because increase in the flow rate increases theReynolds number, which in turn increases the Stantonnumber and thereby the film heat transfer coefficient.The increase in film heat transfer coefficient willincrease the overall heat transfer coefficient. This willalso cause a decrease in the tube outlet temperature, ascan be observed form tables. This is because increase inthe volumetric flow rate increases the mass flow rate ina much faster rate than over all heat transfer coefficientor the heat energy transferred. Since the specific heatremains almost constant, tube outlet temperature shoulddecrease to comply with law of conservation of energy.

2) Overall heat transfer coefficient for S and T HE

As the volumetric flow rate of the tube side fluid is increased from 120 to 720 lph, the overall heat transfer coefficient increased from 126.167 to 150.15 W/m^2K. For the same volumetric flow rates, the simulated values varies from 121.805 to 148.605 W/m^2K respectively, i.e., almost same as experimental values.

3) Shell outlet temperature for S and T HE:

For the flow rate increments from 120 to 720 lph, the outlet temperature of the shell side fluid varied from 45 to 31ºC, whereas the simulated values were 42 to 30ºC, respectively.

4) Tube outlet temperature for S and T HE:

For the flow rate increments from 120 lph to 720 lph, the outlet temperature of tube side fluid varied from 71 to 61.5ºC, whereas the simulated values were 68 to 60ºC respectively.

The results for the other compositions were similar to that obtained from the one considered here as the reference. From the above comparisons it can be said that the mathematical model developed for the system is very close.

C. Conclusion

Many Experiments were conducted on a 1-1 Shell and Tube heat exchanger with different cold side flow rates and different compositions of cold fluid. The effect of these parameters on the shell outlet temperature, tube outlet temperature and overall heat transfer coefficients were studied. It was found that cold fluid outlet temperature decreases and the overall heat transfer coefficient increases with increase in flow rate of cold fluid. Also the outlet temperature of cold fluid decreases and overall heat transfer coefficient increases with increase in composition of water.

The overall effectiveness of heat exchanger was found to increase with decrease in composition of water. It was found that the Cross Flow Heat Exchanger is the most effective compared with the Shell and Tube Heat Exchanger. A mathematical model of this system is developed, simulated using MATLAB and compared with the experimental values. Finally a correlation for the calculation of film heat transfer coefficient is developed using dimensional analysis for tube side.

Mohammed Rabeeh V [3]Design of shell and tube heat exchanger : The design procedure starts with providing the standard dimensions of length and diameter of the tube which are proposed by tabular exchange manufacturesassociation (TEMA) to the MATLAB code written. Program is run by iterating with possible combination of th standard dimensions and the overall heat transfer coefficient (U) is obtained in each case. The obtained values of the U are compared and the corresponding dimensions for the maximum value fo the same is obtained as the output.

The design procedure is as follows:

Assume the tube length and outer diameter according to TEMA specification.

0.0666 < (Shell diameter/Tube length) < 0.2 (1)

1.25 < (Pitch / outer dia. of tube) < 1.5 (2)

No of tubes = $\pi C_T P D_s^2 / 4 CL P^2$ (3)

After this Reynolds number and then heat transfer coefficient for the tube side is calculated, which is followed by the calculation of these parameter fo the shell side.

1) Tube side :
Cross flow area = (Dia. of tube^2)(Nu. Of tubes) / 4(no of passes) (4)

Re = (mass flow rate)(Diameter of tube)/ (Cross flow area* viscosity) (5)

Nusselt number = Nu = hd_o/ k = 1.86 * (viscosity)$^{0.25}$ (Re * Pr * dia. of tubes/2)$^{1/3}$ (6)

Laminar flow

Nusselt number = $hd_o/k = 0.023 (Re)^{0.8} (Pr)^{0.33} (\mu)^{1.25}$(7)

2) Turbulent flow:
From the equation for the nusselt number, heat transfer coefficient is obtained. Similarly heat transfer coefficient for the shell side is obtained. Since the tubes are present inside the shell equivalent diameter (D_e) for the shell is calculated depending on he configuration of the arrangement of the tube inside the shell. Main configurations which are extensively used are triangular and square, depending on which the CTP and CL values vary. Equations for equivalent diameter corresponding to each configuration is obtained from Perry H Greens hand book.

3) Shell side :
Nusselt no = $hd_o/k = 0.36 (R_e^{0.55})(Pr^{0.33}) (\mu/ \mu_w^{0.14})$ (8)

Baffle spacing = 74(outer diameter of tube)$^{0.75}$ (9)

After obtaining these values overall heat transfer coefficient is calculated using the below given equation :

$$Uo = (\frac{1}{Ao})/((\frac{1}{hiAi}) + \left(\frac{\frac{indo}{di}}{2\pi kL}\right) + (\frac{1}{hoAo})(10)$$

4) Steady state time
Analysis of heat exchanger is done best by considering the energy balance differential equation for shell and tube differently

$$\frac{\delta T}{\delta t} = \frac{\vartheta \delta T}{\delta x} - (\frac{\vartheta 2\pi(Tin-Tout)}{\rho Cpr})(11)$$

$$\frac{\delta T}{\delta t} = \frac{\vartheta \delta T}{\delta x} - (\frac{\vartheta 2\pi(Tin-Tout)}{\rho Cp(r^2shell-r^2tube)}) \quad (12)$$

Solving the above energy balance equation using second order Runge-Kutta method by modeling the shell and tube heat exchanger in MATLAB is done using a finer spiral grid.

B.Jayachandriah& K. Rajasekhar [4]An attempt is made in thispaper is for the Design of shell and tube heat exchangers bymodeling in CATIA V5 by taking the Inner Diameter of shellis 400 mm, length of the shell is 700 mm and Outer diameterof tube is 12.5mm, length of Tube is 800mm and Shellmaterial as Steel 1008, Tube material as Copper and Brass.

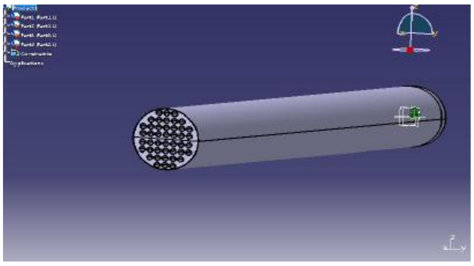


Fig. 3: CATIA model of problem

By using modeling procedure Assembly Shell and Tube withwater as medium is done. By using ANSYS software, thethermal analysis of Shell and Tube heat exchangers is carriedout by varying the Tube materials. Comparison is madebetween the Experimental results, ANSYS. With the help ofthe available numerical results, the design of Shell and Tube heat exchangers can be altered for better efficiency.

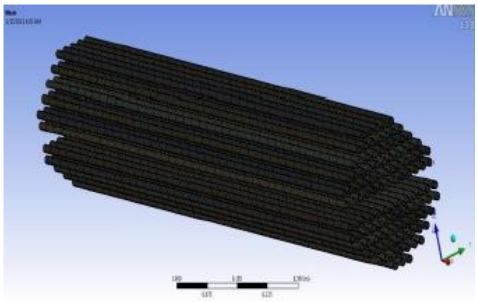


Fig. 4: ANSYS model of problem

Dimensions of Modelling

No of tubes = 44

Length of the tubes = 800mm

Tube diameter = 25mm

Tube pitch = 32mm

Clearance = Pt – do =32 – 25 =7mm

Tube layout = 90

Shell length = 800mm

Shell diameter = 135

Thickness =9mm.

Thermal properties of steel 1008

Thermal conductivity = 45 W/m °C

Density = 7872 kg/ m3

Specific heat = 481 J/ KG °C

Thermal properties of fresh water

Thermal conductivity = 0.604 W/m °C

Density = 997.4 kg/ m3

Specific heat = 162 J/ KG °C

Thermal properties of Brass

Thermal properties of fresh water

Thermal conductivity = 111 W/m °C

Density = 8600 kg/ m3

Specific heat = 162 J/ KG °C

Thermal properties of fresh water

Thermal conductivity = 400 W/m °C

Density = 8933 kg/ m^3

Specific heat = 385 J/ KG °C

D. Analytical Analysis

General formula for calculating heat transfer:

The heat release from the shell and tube heat exchangerswas obtained by multiplying over all heat transfer co-efficient,Area of tubes and difference of temperatures.

Q= $UA\Theta_m$

Where, A = area of tub,

U = Overall heat transfer coefficient

Area of the tubes = A = $\pi d_o L$.

where, d_o = out side diameter of tubes

L = length of the tubes.

LMTD method:

$$\theta m = LMTD = \frac{(t_{h1} - t_{c1}) - (t_{h2} - t_{c2})}{IN\left(\frac{t_{h1}-t_{c1}}{t_{h2}-t_{c2}}\right)}$$

t_{h1}=hot water inlet

t_{h2}=hot water outlet

t_{c1}=cold water inlet

t_{c2}=cold water outlet

For copper:

Heat release : Q= $UA\Theta_m$

Overall heat transfer coefficient U:

$$\frac{1}{U} = \frac{1}{h_i} + \frac{1}{h_o}$$

1/u = 1/0.604+1/1400

U = 0.603 w/m^2 k

A= 0.05494 m^2

$\Theta_{m =}$ 33.7 K

Heat release : Q = 11.7 W.

For Brass :

Overall heat transfer coefficient U= 0.938 w/m^2 k

Area = A = 0.05494 m^2

LMTD = 27 k

Heat release Q = 8.34 w.

Tabular column of Results (In °K)			
Temperatures		**Copper**	**Brass**
Max. Temp.	Analytical		333
	Hot water	333	333
	Cold water	310	308
Min Temp.	Analytical		312
	Hot water	316	310
	Cold water	303	303

E. Discussion

Thus the ANSYS Results is calculated for copper and brass materials. The heat released from copper material is 15.63w and that from brass is 12.98 w which is less when compared with the copper. Whereas literature experimental results are 14.17W for Copper and 11.34W for Brass. Therefore, range of variation is within 10% .Hence Results are validated. The temperature distribution profile of whole assembly in the sectional front view .From the figure it is seen that the maximum and minimum temperature for copper is from 316 K to 310 K and for brass is 318 and 308 which is simulated from the ANSYS Workbench result. Hence, it is observed that copper gives better heat transfer rate compared with that of brass.

F. Conclusion & Future Work

After performing all the analysis work for shell & tube heat exchangers the following observation had been done. From the study of the result as mentioned in table 1 , after performing the calculation the fluid water for bass output temperature is 310 °k which is nearer to the value mentioned in output temperature of ANSYS. As we change the tube material from the brass to the copper, temperature difference between output temperature of copper & brass had been varied.

Analysis has been done by varying the tube materials and it is found that copper material gives the better heat transfer rates than the brass material.

(1) Rate of heat transfer can be improved by varying the tube diameter, length and no of tubes.
(2) By changing the pitch lay out rate of heat transfer can be improved.
(3) By changing the temperature of tubes and medium rate of heat transfer can be improved.
(4) By changing the materials of tubes heat transfer rate can be improved.

A.O. Adelaja, S. J. Ojolo and M. G. Sobamowo[5] considered both the thermal and mechanical design of the E-type shell and tube heat exchanger with the aid of computer programming. It involves developing a simple user-friendly computer program for the heat transfer calculations and ensures that the computational time is kept minimal. The algorithm is designed such that after the conditions for the thermal analysis are satisfied, the program automatically proceeds to the mechanical design. The program written in

Visual Basic was tested using a model and the simulated result presented. Software has been developed for the thermal, hydraulic and mechanical design of a shell and tube heat exchanger using the Kern model[16] to evaluate the coefficient of heat transfer and the pressure drop. The interactive nature of the user-friendly, object oriented Visual Basic software enables it to first do the thermal-hydraulic design after which it proceeds to the mechanical module when all conditions are satisfied. Also changes in the conditions, parameters and geometry can be easily effected at any point in the design process. Drawings however can only be made after the mechanical design and not before it. The software will be useful for both industrial applications and educational purposes.

Yusuf Ali Kara and OzbilenGuraras [6] prepared a computer based design model for preliminary design of shell and tube heat exchangers with single phase fluid flow both on shell and tube side. The program determines the overall dimensions of the shell, the tube bundle, and optimum heat transfer surface area required to meet the specified heat transfer duty by calculating minimum or allowable shell side pressure drop. He concluded that circulating cold fluid in shell-side has some advantages on hot fluid as shell stream since the former causes lower shell-side pressure drop and requires smaller heat transfer area than the latter and thus it is better to put the stream with lower mass flow rate on the shell side because of the baffled space.

Rajagapal Thundil KaruppaRaj and SrikanthGanne [7] made the attempts to investigate the impacts of various baffle inclination angles on fluid flow and the heat transfer characteristics of a shell-and-tube heat exchanger for three different baffle inclination angles namely 0°, 10°, and 20°. The simulation results for various shell and tube heat exchangers, one with segmental baffles perpendicular to fluid flow and two with segmental baffles inclined to the direction of fluid flow are compared for their performance. The shell side design has been investigated numerically by modelling a small shell-and-tube heat exchanger. The study is concerned with a single shell and single side pass parallel flow heat exchanger. The flow and temperature fields inside the shell are studied using non-commercial computational fluid dynamics software tool ANSYS CFX 12.1. For a given baffle cut of 36%, the heat exchanger performance is investigated by varying mass flow rate and baffle inclination angle. From the computational fluid dynamics simulation results, the shell side outlet temperature, pressure drop, re-circulation near the baffles, optimal mass flow rate and the optimum baffle inclination angle for the given heat exchanger geometry are determined. It was concluded that the shell and tube heat exchanger with 20° baffle inclination angle results in better performance compared to 10° and 0° inclination angles.

S. NoieBaghban, M. Moghiman and E. Salehi [8] analyzed the thermal behavior of the shell-side flow of a shell-and-tube heat exchanger using theoretical and experimental methods. The experimental method provided the effect of the major parameters of the shell-side flow on thermal energy exchange. In the numerical method, besides the effect of the major parameters, the effect of different geometric parameters and Reynolds no (Re) on thermal energy exchange in shell-side flow has been considered. Numerical analysis for six baffles spacing's namely 0.20, 0.25, 0.33, 0.50, 0.66, and 1.0 of inside diameter of the shell and five baffle cuts namely 16%, 20%, 25%, 34%, and 46% of baffle diameter, have been carried out. In earlier numerical analyses, the repetition of an identical geometrical module of exchanger as a calculation domain has been studied. While in this work, as a new approach in current numerical analysis, the entire geometry of shell-and-tube heat exchanger including entrance and exit regions as a calculation domain has been chosen. The results show that the flow and heat profiles vary alternatively between baffles. A shell-and-tube heat exchanger of gas-liquid chemical reactor system has been used in the experimental method. Comparison of the numerical results show good agreement with experimental results of this research and other published experimental results over a wide range of Reynolds numbers (1,000-1,000,000).

A.GopiChand et.al. [9]has proposed a simplified model for the study of thermal analysis of shell-and-tubes heat exchangers of water and oil type is proposed. Shell and Tube heat exchangers are having special importance in boilers, oil coolers, condensers, pre-heaters. They are also widely used in process applications as well as the refrigeration and air conditioning industry. The robustness and medium weighted shape of Shell and Tube heat exchangers make them well suited for high pressure operations. This paper shows how to do the thermal analysis by using theoretical formulae and for this they have chosen a practical problem of counter flow shell and tube heat exchanger of water and oil type, by using the data that come from theoretical formulae, they designed amodel of shell and tube heat exchanger using Pro-E and done the thermal analysis by using FLOEFD software and comparing the result that obtained from FLOEFD software and theoretical formulae. For simplification of theoretical calculations they have also done a MATLAB code which is useful for calculating the thermal analysis of a counter flow of water-oil type shell and tube heat exchanger. The result after comparing both was that they were getting an error of 0.023 in effectiveness.

USMAN UR REHMAN [10] an un-baffled shell-and-tube heat exchanger design with respect to heat transfer coefficient andpressure drop is investigated by numerically modeling. The heat exchanger contained 19 tubesinside a 5.85m long and 108mm diameter shell. The flow and temperature fields inside the shelland tubes are resolved using a commercial CFD package considering the plane symmetry. Aset of CFD simulations is performed for a single shell and tube bundle and is compared withthe experimental results. The results are found to be sensitive to turbulence model and walltreatment method. It is found that there are regions of low Reynolds number in the core of heatexchanger shell. Thus, k_! SST model, with low Reynolds correction, provides better results ascompared to other models. The temperature and velocity profiles are examined in detail. It is seenthat the flow remains parallel to the tubes thus limiting the heat transfer. Approximately, 2/3rdof the shell side fluid is bypassing the tubes and contributing little to the overall heat transfer.

Significant fraction of total shell side pressure drop is found at inlet and outlet regions. Due to

The parallel flow and low mass flux in the core of heat exchanger, the tubes are not uniformly heated. Outer tubes fluid tends to leave at a higher temperature compared to inner tubes fluid.Higher heat flux is observed at shell's inlet due to two reasons. Firstly due to the cross-flow and secondly due to higher temperature difference between tubes and shell side fluid. On the basisof these findings, current design needs modifications to improve heat transfer.

G. Literature summary

Many researchers worked on shell and tube type heat exchanger particular based on design, geometrical shape, changed different parameters like various temperature, mass flow rate, pressure for obtaining better heat transfer rate. They have concluded that all above things by experimental and theoretical point of view but few researchers worked on CFD analysis and compared with experimental data. Present days CFD analysis is too much well known for the optimum heat transfer rate in heat exchanger but nobody aware about that analysis and this is reduced the cost of research.

Based on the various studies mentioned in the research papers it is observed that the optimization of the performance and design of shell and tube type heat exchanger can be done by changing the process parameters:

Following are the few guidelines for optimization of heat exchanger performance:

(1) we can utilize the pressure drop margin for optimization as the velocity increases the pressure drop increases but the same time the overall heat transfer coefficient will be increased and maximum heat transfer can be achieved.
(2) Changing in tube metallurgy will also affect the overall heat transfer coefficient. Higher the thermal conductivity higher the overall heat transfer coefficient.
(3) Change in number of passes will increase the velocity will in turn more hat transfer.
(4) Change in baffle geometry will give more heat transfer.

While designing it tis must to take care of all other parameters like pressure drop, as it may be possible that the heat transfer will increase but the power requirement for the pumping will increase that will lead to increase in operating cost. It may also be taken in account that the maintenance ease and cost to be considered during shell and tube type heat exchanger , I.e. triangle pitch will be tough to clean out.

Rating and sizing of shell and tube type heat exchanger requires a lot of time and cost due to manual calculation and experimental model. So, the simulation of the model is the easiest and cheaper way for Rating and sizing of heat exchanger. By using software simulation technique the design of shell and tube type heat exchanger can be done very effectively in very short time period.

REFERENCES

Journal Papers:

[1] Ebieto, C.E. and Eke G.B., "Performance Analysis of Shell and Tube Heat Exchangers using Miscible System: A case study", *Journal of Emerging Trends in Engineering and Applied Sciences,2012 3 (5), pp. 899-903.*

[2] M. Thirumarimuegan, "Performance analysis of shell and tube type heat exchanger using Miscible system", *American journal of applied science, 2008, 5(5): 548-552.*

[3] Mohammed Rabeeh V., "Design of shell and tube heat exchanger Study on the heat transfer and flow sing MATLAB and finding the steady state time using energy balance equation", *International Journal of advance Mechanical Engineering Volume 4, Number 1 (2014), pp. 95–100.*

[4] B Jayachandriah1, K. Rajsekhar, "Thermal Computer Aided Analysis of Thermal and Mechanical design of shell and tube exchangers,"*Advanced materials vol. 367(2012) pp 731-73, Trans Tech publications, switzerland.*

[5] A.O. Adelaja, S.J. Ojolo and M.G. Sobamowo, "Computer Aided Analysis of Thermal and Mechanical Design of Shell and Tube Heat Exchangers", *Advanced Materials Vol. 367 (2012), pp731-737, Trans Tech Publications Switzerland*

[6] Yusuf Ali Kara, OzbilenGuraras, "A computer program for designing of shell and tube heat exchanger", *Applied Thermal Engineering 24(2004) 1797-1805.*

[7] RajagapalThundilKaruppa Raj, SrikanthGanne, "Shell side numerical analysis of a shell and tube heat exchanger considering the effect of baffle inclination angle on a fluid flow", *ThundilKaruppa Raj, R., et al: Shell side numerical analysis of shell and tube heat exchangers, Thermal science: Year 2012, Vol.16, No.4, pp 1165-1174.*

[8] S. NoieBaghban, M. Moghiman and E. Salehi, "Thermal Analysis of shell side flow of shell and tube type heat exchanger using experimental and theoretical methods ", *Institute for Scientific Research, University of Guanajuato Lascurain de Retana No. 5, 36000 Guanajuato, Gto, México*.

[9] AGopichand, Prof.A.V.N.L. Sharma, G. Vijay kumar, A. Srividya, "Thermal analysis of shell and tube type heat exchanger using MATLAB and FLOEFD software.", *Volume 1 Issue :3, ,pp 279-281.*

[10] Usman UR Rehman, "Heat transfer optimization of shell and tube heat exchanger through CFD studies ", *Master thesis in innovative and sustainable Chemical engineering from Chalmers University of Technology*.