VOL. 03: NOVEMBER, 2015 ISSN 2222-1247

DESIGN, EVALUATION AND OPTIMIZATION OF SHELL AND TUBE HEAT EXCHANGER INSTALLED AT MASOOD TEXTILE MILLS, FAISALABAD

Dr. Syed Amjed Ahmad*, Engr. Umair Munir, Engr. Nahid Javed
NFC Institute of Engineering & Fertilizer Research, Faisalabad
Correspondence Authors: amjad.ahmad@gmail.com

Abstract: The Design and Optimization of shell and tube heat exchanger requires great attention in order to develop a heat exchanger which can accomplish its desired task efficiently. This design of heat exchanger design was to minimize the overall cost as much as possible while making no compromises on the exchanger's heat duty. Hence the estimation of the minimum heat transfer area, required for a given heat duty was of utmost importance as it governed the overall cost of the heat exchanger. The Project was to solve the major problem faced by industry i.e. the overheating of Gas Engine Jenbacher JGS 320 GS-N.L installed at Masood Textile Mills. The Shell and Tube Heat Exchanger was used to cool the lubricating oil which was to optimize and to overcome the overheating problem. The Optimization techniques included, changing the flow direction, flow rates, varying the baffle cut and baffle spacing, using finned tubes and the combination of these methods were designed and evaluated and then best method with maximum mean thermal efficiency and required temperature changer was selected as a solution of problem.

Index Terms: Heat exchanger, MTM (Masood Textile Mills, Faisalabad), Overall Heat Transfer Coefficient, ANSYS Fluent CFD, LMTD.

1. Introduction

In process industries, the heat exchangers are one of the mostly used equipment. Heat between two process streams is transferred by means of a heat exchanger. Any process which involve cooling, heating, condensation, evaporation or boiling will require heat exchanger for these purpose. Shell and tube heat exchangers of various sizes play a vital role in industrial operations and energy conversion systems. Tubular Exchanger Manufacturers Association (TEMA) regularly publishes the standards and designs recommendations for shell and tube heat exchangers [1]. Using the TEMA standards and recommended correlation based analytical approach, Shell and tube heat exchanger has been designed very successfully. If at a given iteration, the performance of the considered design is unsatisfactory, then a better performing design can also be obtained by changing the design parameters in right direction. An experienced heat exchanger designer knows what to change in what direction. For example: If the tube side heat transfer coefficient comes out smaller than expected then the designer can guess that this is due to low velocities and then try a configuration with a smaller cross-sectional flow area in next iteration. Although it is relatively simple to adjust the parameters of tube side as compared to the shell side. An ability to visualize the flow and temperature fields on the shell side can simplify assessment of the weaknesses, thus directs the designer to the right direction. Computational Fluid Dynamics (CFD) can be very useful to gain that ability.

The Gas Engine Jenbacher JGS 320 GS-N.L installed at Masood Textile Mills, was overheating due to overloading and the problem could be reduced if the temperature of lubricating oil is maintained in the shell and tube heat exchanger used to cool the lubricating oil with the help of jacket water flowing in tubes and lubricating oil surrounding the tubes in the shell.

In this study, the heat exchanger installed at industry was first designed mathematically The most commonly used correlation based approaches for designing the shell side are the Kern method [2] and the Bell-Delaware method [3]. Bell-Delaware is a Comprehensive method to design shell and tube heat exchanger. The boundary conditions required for designing the heat exchanger using Bell-Delaware method were obtained and the heat exchanger was designed mathematically. The Specification data sheet of Heat Exchanger was obtained [4]. The design was verified by ASPEN Plus software which is a Process designing software. ANSYS simulation software enables us to confidently predict how the product will operate in the real world. The Geometry design for CFD simulation was designed using the SolidWorks which is a complete 3D modeling tool.
After the design is obtained and evaluated with the help of two reliable software, and the heat exchanger installed at the industry was now on our computers and every change in it would show the effect in the real exchanger, so the next process of optimization was carried out.

Two important problems in heat exchanger analysis are

1. Rating existing heat exchangers
2. 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 complexity with experimental techniques involves quantitative description of flow phenomena using measurements dealing with one quantity at a time for a limited range of problem and operating conditions. Computational Fluid Dynamics is now an established industrial design tool, offering obvious advantages. In this study, a full 360° CFD model of shell and tube heat exchanger is considered. By modelling the geometry as accurately as possible, the flow structure and the temperature distribution inside the shell are obtained. (5)

2. Material & Method

The governing equations solved by FLUENT and the turbulence models used for this simulation are explained. Two equation models are used for the simulations. Flow equations and energy equations are described in detail. The wall treatment methods are also discussed and how they are important for modeling the heat transfer is also described.

Flow calculation is governed by the continuity equation, the energy equation and Navier-Stokes momentum equations.

The continuity equation describes the conservation of mass and is written as in equation

[illegible]

Momentum Equation (Navier-Stokes Equation) is

[illegible]

o Energy Equation:

Energy is present in many forms in flow i.e. as kinetic energy due to the mass and velocity of the fluid, as thermal energy, and as chemically bounded energy. Thus the total energy can be defined as the sum of all these energies.

$$h = hm + hT + hC + \omega$$

Kinetic energy

$$hm = 1/2\, \rho UiUi$$

Thermal energy

$$hT = \sum n\, m\_n \int\_Tref^{\wedge}T\; [Cp.ndT]$$

Chemical energy

$$hc = \sum n\; m\_n\; h\_n$$

Potential energy

$$\omega = g\_i\; x\_i$$

o Turbulence Model:

The RANS models assume that the variables can be divided into a mean and fluctuating part. [6]

The pressure and velocity are then expressed as:

$$U\_i = (U\_i) + ui$$

$$P\_i = (P\_i) + pi$$

Where the average velocity is defined as:

$$U\_(i = 1/2T \int\_(-T)^{\wedge}T\; [(U\_i)] \,)\partial t$$

The decomposition of velocity and pressure inserted into Navier-Stokes equations gives:

[illegible]

o Two Equation Model:

[illegible]

And for [illegible]

[illegible]

Where,Qt= overall heat required KW, A= area of dryer, ΔTm = mean temperature difference 0C

For counter flow arrangement,

$$\Delta Tm = \frac{(Tgi - Tmo) - (Tgo - Tmi)}{Ln\ (Tgi - Tmo)/\ (Tgo - Tmi)}$$

o Standard Wall Functions

The average velocity in the interior region of the boundary layer can be devised on the general form in Equation

$$〖(U)〗^+ = f\ 〖(y)〗$$

Assuming that the total stress is constant and the turbulent part of the total stress tensor is negligible in the viscous sub layer,

$$T\_w/p = v\ (d(U\_x))/dy$$

Integrating with respect to y and applying the no slip boundary conditions gives

$$〖(U〗\_x) = Tw Y/pw = U\_(w^2 y)/v$$

Or in the dimensionless form

$$〖(U_x)〗^+ = 〖(y)〗^+$$

In the completely turbulent layer, the total stress tensor shrinks to T_xy=-(UxUy) As the shear stress is almost constant over the inner region of the boundary layer and is approximately equal to T, we obtain

$$T\_w = -p(UxUy)$$

By introducing Prandtle's mixing length model and the relation, l=ky we obtain [7]

[illegible]

3. Results & Discussions

The mathematical design of shell and tube heat exchanger was evaluated and results were verified using ASPEN Plus and ANSYS CFD. The optimization techniques could now be applied to the heat exchanger and its results are analyzed and compared in this chapter. The Mathematical Design of the Optimization had been successfully completed using Bell Delaware method and TEMA Standards considering the given boundary conditions.

The Process simulation of the designed geometry was done using ASPEN Plus and the results were same as the original model installed at the industry.

The 3D model of the exchanger was designed using SolidWorks.

The CFD simulation of the designed model was done using ANSYS Fluent 15.0.

The Results of the ASPEN and ANSYS verified that model was same as the original model installed at the industry.

The change of flow from co-current flow to counter current flow had changed the efficiency from 15.26% to 15.32% and the temperature difference had increased 0.03 K only which was not sufficient for solving the overheating problem. The increase in flow rate of cold water although increased the heat duty but on the other hand it also decreased the mean temperature efficiency.

The decrease in baffle cut from 25% to 20% had increased the mean temperature efficiency from 15.26% to 15.86% which was also not sufficient for the overheating problem.

On varying the baffle spacing from D_s/5 toD_s, the maximum mean temperature efficiency is at D_s/3 baffle spacing which was 15.86 % and also the pressure drop in the shell side had decreased from D_s/5 toD_s.

Using the finned tubes instead of bare tubes gave the maximum efficiency which was 26.78 %. The pressure drop in the shell side had also increased but this was not that much to consider.

Using the finned tubes with D_s/3 Baffle Spacing had increased the efficiency further to 27.72 %. The pressure drop in the shell side had also decreased.

Using the finned with D_s/3 Baffle Spacing and 20% baffle cut had further increased the mean temperature efficiency to 28.56 % which was suitable to solve the problem of overheating.

The optimization technique with most efficiency was proposed as a solution to the overheating problem.

The objective of optimization of shell and tube heat exchanger was achieved successfully.

Fig. (1)

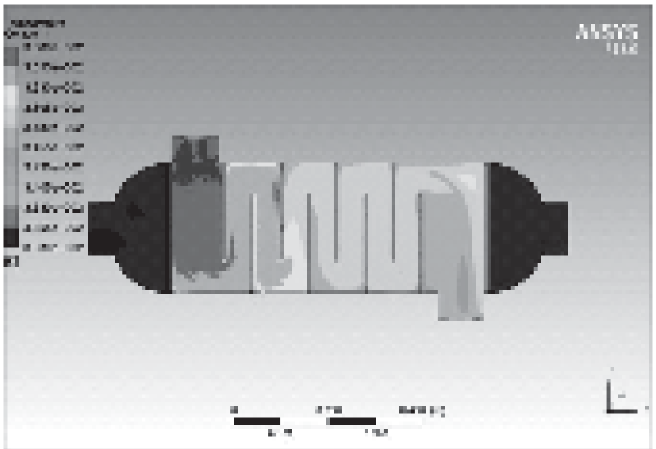

Fig. (2)

References

[1] Standards of the Tubular Exchanger Manufacturers Association. 9th ed. Gaddis D, Ed. Tarrytown, N.Y.: TEMA Inc.; 2007.

[2] Kern DQ. Process heat transfer. New York, N.Y.: McGraw-Hill; 1950.

[3] Bell KJ. Delaware method for shell side design, in heat exchangers: thermalhydraulicfundamentals and design. (Eds) Kakaç S, Bergles AE, Mayinger F.New York: Hemisphere; 1981; 581-618.

[4] Qiuwang Wang, Qiuyang Chen, Guidong Chen, Min Zeng, "Numerical investigation on combined multiple shell-pass shell-and-tube heat exchanger with continuous helical baffles", International Journal of Heat and Mass Transfer 52 (2009).

[5] M.R.Salimpour, "Heat transfer coefficients of shell and coiled tube heat exchangers", Experimental Thermal and Fluid Science 33 (2009) 203–207. [15] K.M. Lunsford, “Increasing Heat Exchanger Performance” ,Bryan Research and Engineering, Inc., Vol. No.1, pp. 1-13, March 1998.

[6] Huadong Li, Volker Kottke ,” Effect Of Baffle Spacing On Pressure Drop And Local Heat Transfer In Shell-And-Tube Heat Exchangers For Staggered Tube Arrangement” ,International Journal of Heat Mass Transfer, Elsevier Science , Germany, 1998.

[7] Huadong Li and Volker Kottke," Effect Of the Leakage on Pressure Drop and Local Heat Transfer in Shell-and-Tube Heat Exchangers for Staggered Tube Arrangement", Inl. J. Heat Mass Transfer, Elsevier Science Ltd., 1998.