Journal of Maritime Research, Vol. V. No. 1, pp. 47-64, 2008

ISSN: 1697-4840

DESIGN AND SIMULATION OF A VIRTUAL TUBULAR HEAT EXCHANGE UNIT FOR EDUCATIONAL APPLICATIONS

E. Eguía[1], A. Trueba[2] and M. M. Milad[3]

Received 24 August 2007; received in revised form 9 September 2007; accepted 22 February 2008

ABSTRACT

One of the tasks of an industrial plant designer is the choice of the most suitable elements that will make up the said plant, so that all the devices complement and are synchronised with each other. If the industrial plant includes several heat exchanger units, each one of these should match its particular working conditions, which will be determined by the work process each exchanger performs. When designing and choosing the most appropriate heat exchanger for a set work process, certain international standards must be observed and likewise the recommendations of the classifying societies and the methods currently used in industry all have to be taken into account. In view of this, and taking into account the most advanced methods in heat exchange unit design, a computer program has been created that enables us to design a virtual heat exchanger that fits each of our needs and on which we can simulate different thermal processes and analyse its behaviour in all kinds of situations.

Keywords: Heat exchanger, design, simulation, software.

INTRODUCTION

In any heat exchange unit calculation, a series of economic factors that are going to have a direct effect on the final performance of the installation must be assessed. It

[1] University Professor, Department of Sciences & Techniques of Navigation and Ship Construction (eguiae@unican.es), Cantabria University, Santander, Spain. [2] Assistant Professor, Department of Sciences & Techniques of Navigation and Ship Construction (truebaal@unican.es), Cantabria University, Santander, Spain. [3] Ph.D. in Marine Sciences (milad@ono.com), Cantabria University, Santander, Spain.

is essential to carry out a study of the variables that have an influence both on heat transfer and on losses of head. Resistance to heat transfer is minimised when working with a high Reynolds Number. This enables a heat exchanger to be projected with a low heat exchange surface area thus reducing the cost of the product. However, a high Reynolds Number leads to an increase in loss of head, thus making it necessary to use an impulse pump with a greater range, hence making the product more expensive (Kreith and Bohn, 2001).

A factor that must be taken very much into account during the design of a heat exchanger unit is the internal fouling of the tubes. Biological deposits adhered to the inside surface of the tubes represent a huge problem both for land and for sea based industry, as they reduce heat transfer and therefore the cycle's thermal performance (Eguía, 1998). Thus, when the time comes to make calculations, the fall in the transmission coefficient over the period when the exchange unit is active will have to be taken into account. Likewise, a differentiation will have to be made in the working conditions depending on whether it is new or if it has already been in service and for how long.

Furthermore, for certain inlet conditions and certain operating fluid velocities, it is necessary to consider the concept of marginal transmission. Heat flow is directly proportional to the mean temperature difference of the operating fluids, and it decreases as the heat exchange surface area rises. Therefore, in the heat exchange unit financial calculation we must take into account that an increase in the surface area corresponding to the marginal transmission implies an additional cost. This cost will have to be assessed versus the increase in calories recovered. Normally we consider economic approximation to the lower difference in temperatures taken at each of the ends (it is necessary to introduce an efficiency factor specific for each kind of exchanger unit into the calculation (Rohsenow et al., 1998)).

It should be emphasized that the optimisation calculation for a heat exchanger is a complex process, as it forms part of a whole installation and cannot be dealt with in isolation. All of this complex calculation process is simplified when simulators are created that enable exact and rapid data to be obtained by taking all the technical and economic imperatives of today's industry into account.

This has therefore led to the creation of a thermal-hydraulic design program for shell and tube heat exchangers like the one shown in Figure 1. It starts from initial data supplied by the client such as the operating fluid inlet and outlet flows and temperatures, and its main features are:

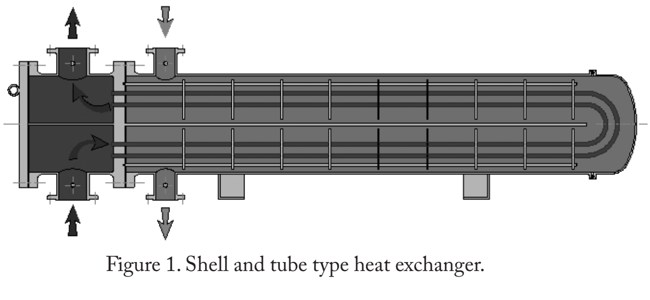

Figure 1. Shell and tube type heat exchanger.

— Improvements in the heat exchanger design and evolution process.
— Ease of calculation of the thermal-hydraulic process and of heat exchange.
— Calculation of the fluid stream convection coefficients.
— Calculation of the overall heat transmission coefficient.
— Calculation of dimensionless values such as the Reynolds Number, Nusselt Number, etc.
— Heat transmission by conduction in multiplayer pipelines and walls.
— Automatic attainment of the properties of certain fluids such as fresh water, salt water and air.
— Calculation of the absolute viscosity of 99 fluids, based on their temperature and density.
— Automatic unit conversion, making them easier to be handled and enabling the user to introduce data in any available unit.

The calculated data may be converted or transformed into the unit favoured by the user.
— Calculation of the thermodynamic properties of water vapour and Mollier diagrams up to 1000ºC.
— A Moody diagram for friction in conduits and pipelines.
— The graph presentations have a real zoom function and are designed for easy reading.
— Working by use of independent windows and calculation modules.
— Data is saved on a standard database, thus enabling it to be transported and shared with other programs.
— It employs modern programming technology and an "intelligent" data introduction module.

The design process is carried out using the method recommended by TEMA (TEMA, 1999), given that the said method may be applied in computer assisted design techniques (Leong et al., 1998).

Once the heat exchanger has been designed, the target is that it fits as good as possible the heat exchange process that it is going to perform in normal real conditions, simulating different operating situations:
— Normal or design conditions in which the heat exchanger works at full capacity and without going into head losses.
— Abnormal conditions with unpredictable situations that have not been taken into account during the calculation and design phase.
— Critical conditions in extreme operating situations before entering into head losses.

For example, in the case of conventional land or sea based thermal plants using seawater as their coolant fluid, the temperature of the said coolant would vary depending on the geographical area and the season of the year. It is therefore necessary to use a simulation program to find out how the heat exchanger will behave before such variations in temperature or to what temperature it will work at an acceptable performance level.

Hence, a simulation program has been created in order to study and predict heat exchanger behaviour in a variety of working conditions. It allows us to (Kotake and Hijikata 1993):

— Simulate functioning in normal conditions while varying environmental parameters within the set margins.
— Study the effect caused by a variation in each of the exchanger parameters and features on the other parameters and environmental conditions.
— Calculate its working range limits.
— Study heat exchanger behaviour in extreme situations and identify potential failures under these conditions.
— Study and analyse behaviour in abnormal conditions that may arise for a short period.
— This simulation module is an interesting tool for studying and learning how a heat exchanger works, obtaining a more reliable and balanced design.

THE HEAT EXCHANGER DESIGN PROCESS

The design of a heat exchanger for a specific process involves handling of a series of data and parameters that are going to determine its features. It is the designer's job to obtain the said data and parameters for the work process for which it is going to be designed. Likewise, it is the designer who should choose the design method to be followed (Minkowycz et al., 1988).

Definition of the Process

The client determines the heat transmission process that is required in his plant. An example process scenario might be a 1–2 shell and tube type heat exchanger to cool 19 kg/s of an organic liquid from 71ºC to 49ºC, using water as the coolant fluid with an inlet temperature of 21ºC. The outlet temperature must not exceed 49ºC. The following aspects would then be set:

— Material of which the tubes are made: steel tubes (14 SWG)
— Outer diameter of the tubes: 19 mm
— Length of the tubes: 243.8 mm
— Arrangement: Triangular and 25.4mm separation between tubes
— The coolant fluid fouling factor: 0.00018 m^2 K/W

The physical properties of the fluid to be cooled also have to be taken into account for the given range of working temperatures:

— Dynamic Viscosity μ = 0.45 centiPoise
— Density ρ = 882 kg/m^3
— Specific Heat c_p = 0.93 kJ/kg K
— Prandtl Number N_{pr} = 4.356

The program automatically calculates the physical properties of a water coolant; therefore it is not necessary to use tables when working with either fresh or salt water.

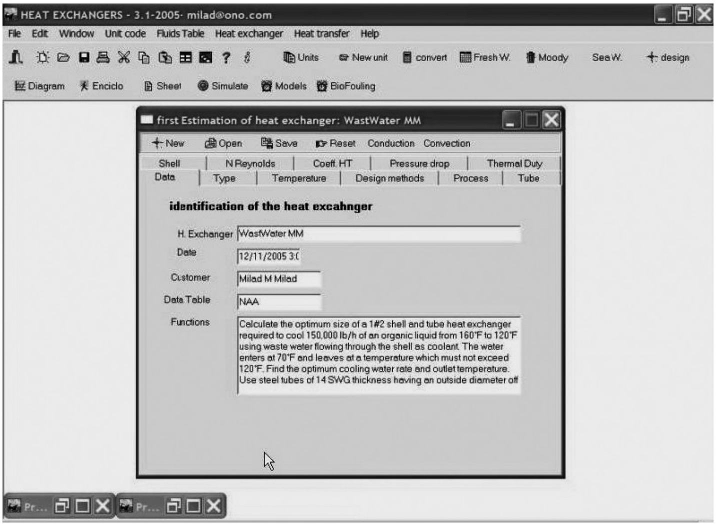

Figure 2. Main window of the design program.

To Create a New Design

The design module is started from the main program window (Figure 2). Access to the heat exchanger primary estimation window is attained by introducing the following variables into their corresponding windows (Kim et al., 2002):

— *Heat exchanger identification data:* Name, date, author and process description. This data is saved on a database. The said database may store different designs, which may be modified whenever required.

— *Choice of the type of heat exchanger identification*: The choice of the heat exchanger and the flow type will depend on the fluids that take part in the process, e.g. liquid-liquid, vapour-liquid, etc.

— *Process temperatures*: Here we introduce the inlet and outlet temperatures and mass flow rates along both the tube side and the shell side.

— *Mean caloric temperatures*: These are necessary to automatically calculate the physical properties of the fluids, the convection coefficients both on the shell side and on the tube side, and also the estimated fouling coefficients must be introduced. Likewise, the properties of the operating fluids, both those inside the tubes and through the shell, are also introduced. If an organic liquid is employed, the user must put this data in manually; however if the coolant fluid is water, its properties are already included in the program. If other fluids are used, their dynamic viscosity can be calculated from their density. The program has a list of 99 fluids available.

— *Calculation of the energy involved in the process:* In the *Process* window we can calculate the heat exchanger effectiveness and the thermal capacity relationship of the fluids. This data is necessary to calculate the effective temperature correction factor. We then go on to calculate the mean logarithmic temperature and the effective temperature. Likewise, the correction factor is calculated by clicking on the "Correction Factor" button and selecting the type of heat exchanger, following the TEMA classification. It has to be borne in mind that the correction factor must be over 0.75. If this is not the case it is advisable to change the configuration. We then go on to calculate the heat flows, the overall heat transmission coefficient and the heat transfer surface area required by the process.

— *Tube dimensions:* If the client supplies this data, it is introduced into the corresponding data cells. However, if no data is supplied, there is a dropdown menu showing tube diameter, pitch and arrangement (Serna and Jimenez, 2004).

— *Shell Dimensions:* First we select the number of shell pitches from the dropdown list (following the TEMA classification), and then we calculate the number of tubes to thus obtain the diameter of the shell. We should also introduce the tolerance between the shell and the tube bundle; this value should lie between 10 and 20 mm. As shell dimensions are standardised, we will have to choose the shell that just exceeds the calculation results obtained. In this section, we also calculate the shell nozzle that will condition the head losses due to the fluid velocity, cavitation and erosion of the tubes. Mass fluid state, density and rate enable us to derive the minimum nozzle diameter, the maximum velocity and the distance from the nozzle to the tube bundle.

— *Calculation of the Reynolds Number:* Firstly, we calculate the pitch section of the tube bundle, i.e. the sum of the sections of all the tubes that form the tube bundle and this is treated as if it were a single tube with a section that is the total we have calculated. Then we calculate the wear in the tubes and the Reynolds Number. These values will subsequently be used to obtain the film by convection coefficient in the tubes.

— *The overall heat transmission coefficient:* Due to the temperature difference between the wall and the core of the main stream of the two fluids, we must add the so-called *viscosity factor.* It is therefore necessary to calculate the tube wall temperature and obtain the viscosity of both fluids at this temperature. The program carries out these operations automatically. Furthermore, the option exists for introducing the *fouling factor* into the calculation.

— *Tube wall temperature:* Due to the temperature difference between the wall and the core of the main stream of the two fluids, we must add the so-called viscosity factor. It is therefore necessary to calculate the tube wall temperature and obtain the viscosity of both fluids at this temperature. The program carries out these operations automatically. The tube fouling coefficient is similarly applied.

— *The convection coefficient in the tubes:* This module includes the calculation, amongst others, of the Nusselt Number, the convection coefficient and the pressure drop in the tube bundle. It is also possible to carry out flow and heat transmission analyses in tubes that are independent of the heat exchanger as a whole. We must begin by selecting the type of configuration to be analysed from the three options available: The first option, "pipelines and conduits" is suitable for analysing and calculating the flow in an isolated pipeline in which fluid flows and there is a heat energy exchange with the outside. In this case, there is a drop down list for the user to select the option he requires from the various calculation formulae, depending on the pipeline and flow characteristics and his particular preferences. The second option, "tube bundle", refers to the calculation for the heat exchanger as a whole: To calculate the convection coefficient we must go through the following steps:

– The properties of the fluid inside the tubes are derived; both at mean and wall temperatures.
– The calculations can be made *step by step* or in *automatic* mode.
– The fluid velocity inside the tubes is calculated.
– The mean temperature inside the tube is calculated.
– The Nusselt Number is calculated.
– The convection coefficient is calculated.

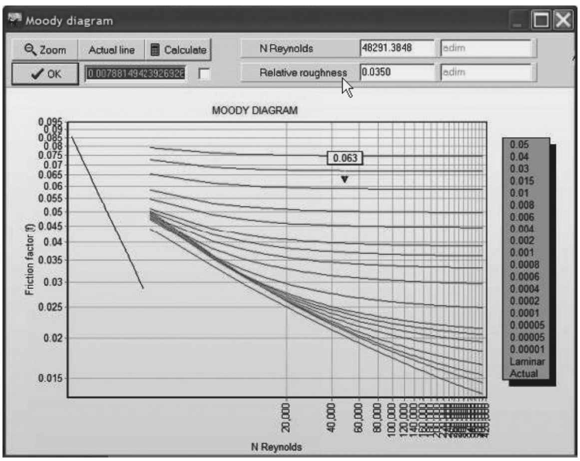

Figure 3. Moody diagram showing the current work point, the relative roughness lines and the friction factor.

The Moody diagram enables us to obtain the pressure drop in the tube bundle. Setting the relative roughness value, the friction factor is calculated and when we close the "Moody" window, the pressure drop in the tubes is calculated automatically. The Moody Diagram module (Figure 3) shows the point we are currently at and the relative roughness lines together with the results of the friction factor calculation.

On concluding the calculation and closing the convection in the tubes module window, we go back to the design window that now shows the results in their corresponding data cells.

— *Calculation of the flow parameters and the shell deflectors:* The profile of the shell fluid stream between the deflectors is set as a combination of cross, counter-current and parallel flows. The calculation of the deflectors is a process that directly affects heat exchanger performance; therefore an independent module has been created to calculate and design all the shell flow properties. We have to go through the following steps:

- The calculations can be made step by step or in automatic mode.
- If the system of units is incompatible, the program automatically undertakes the conversion.
- The "*basic data*" window shows the values calculated in other sections that are necessary here, and it is here that we choose the type of arrangement of the tube bundle (triangular, square, at 30º or at 45º).
- In the "*J Factors*" window the following values are calculated: The transverse area or the equivalent diameter which corresponds to the section the fluid passes through in the shell; the minimum tolerance required between the shell and the flow deflectors obtained from the tables; the maximum space between the deflectors; the tolerance between the tubes and the deflectors, the value of which may range from 5 to 10 mm; and the shell film coefficient correction factors, "J_c, J_L, J_B, J_r, J_s", which are involved in the calculation of the convection coefficient of the outside of the tube bundle. These are calculated by the program.

The "*Shell film coefficient*" window shows the parameters that are automatically calculated by the program and we can obtain the physical properties of the shell fluid by clicking on "*Shell fluid properties*". Calculations are also made for the shell wear, the Reynolds Number, the total J factor expressed numerically or in graph form versus the Reynolds Number and the shell film coefficient.

— The "*Pressure drop*" window serves to calculate the pressure drop correction factors, "R_L, R_b, R_s", and the pressure drop in a bank of ideal tubes to which we will later add the most suitable correction factor for the heat exchanger to be designed. We can also obtain a graph representation of the friction coefficient versus the Reynolds Number. The pressure drops in the shell are then calculated.

— *Thermal balance and design verification:* The aim of this module is to check that the caloric energy exchanged complies with the safety margin on the real process. The safety coefficient is a function of the fouling factor admitted by the heat exchanger before losses imply working beyond the design point. If the safety margin is small, the downtime programmed for its cleaning will be more frequent and the running periods will be shorter, whereas if the safety margin is too large we will have an oversized heat exchanger for the process to be undertaken. Therefore, the ideal situation is that the safety coefficient be close to the expected fouling factor.

Once the heat exchanger design has been concluded, we will only need to print off the data obtained and send this to the manufacturing firm.

AUXILIARY MODULE

In the event of operating with water vapour, an auxiliary module is available that shows the work line depending on the vapour properties in the Temperature-Entropy (T-s) diagram, the Enthalpy-Entropy (h-s) diagram and the Pressure-Enthalpy (p-h) diagram. It is also possible for the user to create his own vapour tables.

THE SIMULATION PROGRAM

The design program allows the user to simulate and analyse the calculated exchanger or alternatively an already existing exchanger whose operation the user wants to analyse.

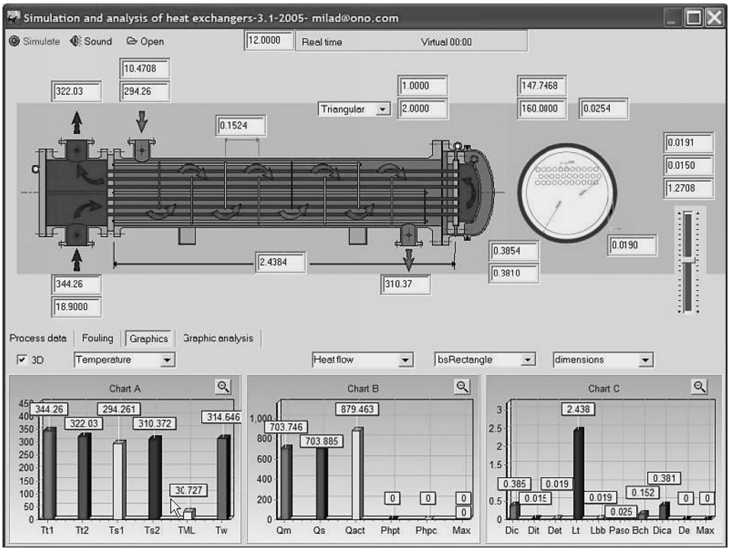

Figure 4. Main window of the simulation and analysis module.

A Description of the Program

In the top part of the main heat exchanger simulator and analysis module shown in Figure 4, there is a shell-tube type heat exchanger unit with its corresponding inlet and outlet nozzles. There are several boxes superposed on the drawing showing data regarding some of the features of the thermal process and the heat exchanger dimensions. The inlets of the cooled and coolant fluids show their temperature and rate of mass flow and the outlets indicate the temperature of the fluids. It likewise shows the dimensions of the exchanger: Tube length, the tube's internal and external diameter, pitch between the tubes and the tube diameter/pitch relationship, the total number of tubes, the shell diameter, arrangement of the tubes on the tube plate, the space between deflectors and the number of tube and shell pitches.

To access additional information for each parameter, we only need to click on the data cell to call up a more complete information box.

The bottom part shows a box of several superposed pages that give us information about:

— *Heat exchange process data.* In this section we can obtain the following numerical data: the physical properties of the operating fluids (their viscosity, density, specific heat, Prandtl Number and thermal conductivity); flow parameters (mass flow rates, velocities, wear and Reynolds Number); and general information concerning the thermal process (logarithmic mean temperature, correction factor, thermal efficiency, current and required area for heat transmission and a list of thermal capacities).

— *Fouling.* We are shown the general outcomes of the simulation: film coefficients, thermal energy transmitted by each fluid, current overall coefficient and the overall coefficient required by the thermal process under consideration, fluid velocities, cutting forces, resistance to fouling, the friction factor, pressure drop and pump force necessary for each fluid to circulate. As the program automatically determines the friction factors, we must access the Moody diagram module to find these results.

— *Graphs.* This section shows the outcome of the simulation and the exchanger dimensions and parameters in graph form, showing us in real time the variation of any parameter of the simulated heat exchange process. The graphs are ordered by variables (temperature, pressure, etc.) and can be selected using a dropdown menu.

— *Graph Analysis.* The graph analysis window offers us a graphic representation of the evolution of any parameter in relation to another parameter. We are able to simulate an infinite number of heat exchangers for a given thermal process to thus determine which is the most appropriate for that particular process.

RESULTS

Design Module

Once the client has defined the process and we have followed the abovementioned design module steps, the data obtained is tabulated so that the manufacturing firm has all the necessary parameters in design order to build the heat exchanger. An example of this is shown in Table 1 for a shell with an external diameter of 43.2 cm and an outlet temperature of 49ºC.

Tube Inlet Temperature	21	ºC

PARAMETER	RESULT	UNIT
Tube Outlet Temperature	49	ºC
Tube Mass Flow Rate	6	kg/s
Shell Inlet Temperature	71	ºC
Shell Outlet Temperature	49	ºC
Shell Mass Flow Rate	19	kg/s
DTML Logarithmic Mean Temperature	24.896	K
Effective Temperature Correction Factor	0.94969016	dimensionless
Effective Temperature	23.64	ºK
Tube Heat Flow	703.46	KW
Shell Heat Flow	703.46	KW
Estimated Overall Heat Transmission Coefficient	1,059	W/m^2K
Heat Transfer Area	28.091	m^2
Capacity Relation	0.799	dimensionless
Effectiveness	0.555	dimensionless
External Tube Diameter	19	mm
Internal Tube Diameter	15	mm
Tube Pitch	25.4	mm
Tube Length	243.8	cm
Number of Tube Pitches	2	dimensionless
Space between Reflectors	15.2	cm
Number of Tubes	192.5	dimensionless
Number of Shell Pitch	1	dimensionless
Internal Shell Diameter	419	mm
Mean Tube Temperature	35	ºC
Mean Shell Temperature	60	ºC
Pitch Section (Bundle)	181	cm^2
Mass Velocity (Tube)	1,041	$kg/s\ m^2$
Reynolds Number (Tube)	34.706	dimensionless
Pitch Section (Shell)	17.897	mm^2
Mass Velocity (Shell)	339.120	$kg/s\ m^2$
Equivalent Diameter	0	mm
Reynolds Number (Shell)	8.830	dimensionless

PARAMETER	RESULT	UNIT
Wall Temperature (Clean Wall)	45	ºC
Bundle Viscosity (Clean Wall)	0.45	centiPoise
Bundle Film Coefficient	11.617	$W/m^2\ K$
Shell Viscosity (Clean Wall)	1	centiPoise
Shell Film Coefficient	1.763	$W/m^2\ K$
Clean Overall Coefficient	1.478	$W/m^2\ K$
Estimated Bundle Film Coefficient	2.840	$W/m^2\ K$
Estimated Shell Film Coefficient	4.261	$W/m^2\ K$
Bundle-Shell Tolerance	19	mm
Friction (Bundle)	0.0029579	dimensionless
Pressure Increase (Bundle)	1.2	kg/cm^2
Friction (Shell)	0.121986	dimensionless
Pressure Increase (Shell)	0.023	kg/cm^2
Required Heat	703.4648	kW
Required Area	20.124	m^2
Required Transmission Coefficient	991.743	W/m^2K
Tube Conductivity	30	W/mK
Fouling Factor	0.00018	W/m^2K
Shell Nozzle Diameter	6	cm
Density (Shell)	882	kg/m^3
Specific Heat (Shell)	1.675	kJ/kg K
Dynamic Viscosity (Shell)	0.45	centiPoise
Prandtl Number (Shell)	4.356	dimensionless
Conductivity (Shell)	0.1731	W/m K
Density (Tube)	995.264	kg/m^3
Specific Heat (Tube)	4.1725	kJ/Kg K
Dynamic Viscosity (Tube)	0.8321	centiPoise
Prandtl Number (Tube)	5.614	dimensionless
Conductivity (Tube)	0.616	W/m K
Viscosity (Tube)	0.8138	cst
Maximum Shell Nozzle Velocity	2.126	m/s
Minimum Nozzle Diameter	60.4	mm
Nozzle-Bundle Distance	20	mm

Table 1. List of the data obtained for a shell with an external diameter of 43.2 cm and an outlet temperature of 49 ºC.

SIMULATION PROCESS

Once we are aware of the heat exchanger design features and of the parameters of the thermal process that take place therein, we can use the *SIMULATION module* to view the evolution of the different variables in relation to any other variable and thus analyse the behaviour of the heat exchanger under all possible working conditions.

— *Variation of the tube outlet temperature in relation to the flow volume in the shell.* If we introduce a variation interval in the rate of mass flow, after a few seconds of simulation we obtain the graph shown in Figure 5. During the simulation process we can observe how the bar graphs show the evolution of the heat exchanger parameters. We can also vary simulation speed by clicking on the *Speed Bar*, in order to better perceive how the parameters vary. Likewise, these results may be simulated under the effects of fouling. We thus aim to analyse how biofouling affects the operation of the exchanger and its localised effect on a specific parameter. Figure 5 shows the comparative effect of fouling on tube outlet temperature versus the shell mass flow rate in a clean exchanger (green line) and in an exchanger with fouling (purple line). We can observe that when there is biofouling for a resistance of 0.00018 m^2 K/W, there is a 60 % increase in the rate of mass flow, which represents a considerable economic loss. This explains the importance of study and research into fouling and the methods for its elimination, especially in condensers and/or coolers in conventional, nuclear, land or sea-based thermal power stations. It also shows the variation of shell fluid inlet temperature and mass flow rate versus the tube fluid outlet temperature. Figure 5 shows that a variation in the shell mass flow rate leads to an inverse variation in the tube outlet temperature. The red line represents the scenario of a variation in shell fluid inlet temperature. This simulation allows the user to predict how the heat exchanger will work and the point to which it can fulfil its mission without entering into losses.

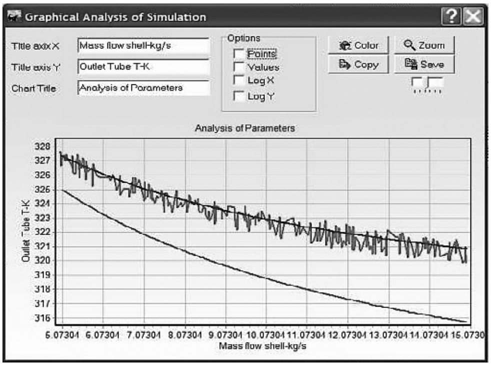

Figure 5. Variation of the tube outlet temperature in function of the mass flow rate in the shell with and without simulation of the effect of fouling.

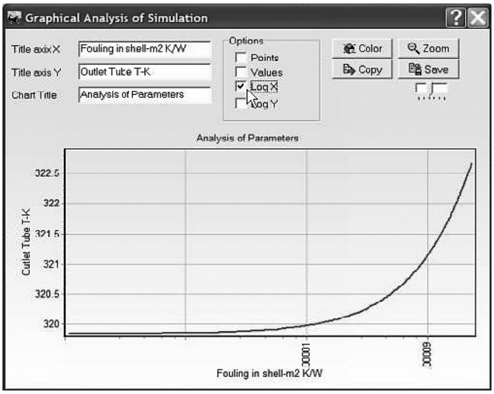

Figure 6. Variation of the tube outlet temperature in function of the resistance of the deposited biofouling.

— *Variation of the tube outlet temperature in relation to the biofouling deposited*. Figure 6 shows how the tube outlet temperature increases with the resistance to heat transfer exercised by the biofouling layer. Likewise, we can represent the effect of biofouling on the overall heat transmission coefficient or on the shell outlet temperature.

- *Variation of the mass flow rate and temperature of the fluid to be cooled.* When this occurs the heat exchanger suffers great thermal tension. Thus to avoid the heat exchanger entering into the area of low thermal performance, it is advisable to take these variations into account during its design or when deciding on the most suitable model. If the maximum mass flow rate of the coolant fluid is constrained by the features of the circulation pump and by the maximum velocity allowed to avoid tube erosion, we need to determine the margins for normal operation and calculate the variations it can bear to consider that it is operating at a satisfactory level. The first step is to calculate the maximum rate of mass flow allowed in the shell. To do so, we simulate a variation in the shell fluid velocity with the mass flow rate, thus attaining the maximum mass flow rate for the maximum velocity allowed that assures no damage will be done to the tubes. The variation in tube temperature versus flow volume or inlet temperature is simulated in the same way. The behaviour of the heat exchanger proves highly interesting when mass flow rate and temperature are varied at the same time, as shown in Figure 7. In this simulation we can view the heat exchanger work field when the outlet temperature is kept constant and the mass flow rate and the inlet temperature are varied. We can see that if the target is to maintain fluid outlet temperature constant (316 K/43ºC) the mass flow rate and inlet temperature must not exceed 348 K/75ºC and 20 kg/s or it would go into losses. It shows the heat exchanger limits for a fluid inlet temperature of 70º to 80ºC and a variation in mass flow rate from 15 to 25 kg/s with a maximum coolant flow volume of 21 kg/s.

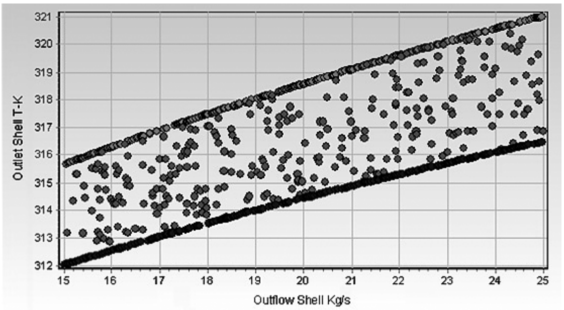

Figure 7. Heat exchanger behaviour when varying the flow volume and the temperature of the fluid to be cooled. Work field.

— *The effect of the tube length on the heat transmitted.* As the exchanger surface area increases more calorific energy is transmitted between the two fluids. In this simulation the tube length varies between 2 and 3 m and the variation on heat transmitted, the tube outlet temperature and the thermal efficiency are all analysed.

The results shown in Figure 8 indicate that the pressure drop in the tube bundle increases as the tubes increases in length. A variation can similarly be simulated in the outlet temperature or in the transmission of calorific energy when there is an increase in the length of the tubes.

— *The effect of the tube arrangement.* Figure 9 shows the variation in shell pressure drop with the variation of the mass flow rate according to how the tubes are arranged on the tube plate (triangle, square or inverse triangle). We can observe that the pressure drop varies according to the tube arrangement; therefore we must bear this factor in mind when working with critical pressure drops. We may likewise do the same with any other variable on the heat exchanger.

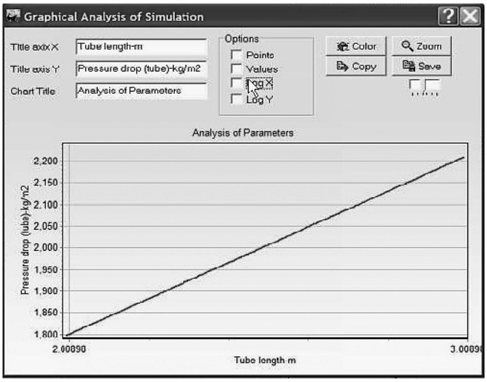

Figure 8. Pressure drop variation in relation to the tube length.

CONCLUSIONS

We have attained a profitable, reliable and didactic tool to assist engineers in industrial plant design and one that fosters an understanding of the processes and mechanisms that take place in heat exchangers. This tool facilitates engineers' work, because it allows them to easily design a heat exchanger, to assess and improve a heat exchanger that has already been designed, to simulate heat exchanger operation in different working conditions, to simulate and analyse biofouling growth and its effects on the designed heat exchanger's performance, as well as to analyse the design method recommended by TEMA.

The design results obtained are comparable to those obtained in real practice in industry, attaining an optimal design that matches the requirements of the heat exchanger's real functions.

A computer simulation of a heat exchanger is a complementary and necessary tool for obtaining an efficient and financially profitable design. It enables us to study the effect caused by a variation in any parameter on the other parameters, on the heat exchanger behaviour and on its performance. Furthermore, the simulations that can be carried out are unlimited and depend only on the user's requirements and imagination.

Figure 9. Variation of the pressure increase in the shell depending on the arrangement of the tubes.

REFERENCES

Eguía, E. (1998). The Problem of Biofouling in Heat-Condensing Interchangers Cooled by Water of Sea. Book Opening of the Academic Course 1998/1999. Cantabria University. Spain.

Kim, J.K., Lee, G.C., Zhu, F. and Smith, R. (2002). Cooling System Design, Heat Transfer Engineering,23(6):49–61.

Kotake, S. and Hijikata, K. (1993). Numerical Simulations of Heat Transfer and Fluid Flow on a Personal Computer. Transport Processes in Engineering, 3, Elsevier.

Kreith, F. and Bohn, M.S. (2001). Principles of heat transfer, Sixth edition, Brooks/Cole, Thomson.

Leong, K.C., Toh, K.C. and Leong, Y.C. (1998). Shell and Tube Heat Exchanger Design Software for Educational Applications, Int. J. Engng Ed. Vol. 14, Nº 3, pp. 217-224.

Minkowycz, W.J., Sparrow, E.M., Schneider, G.E. and Pletcher, R.H. (1988). Handbook of Numerical Heat Transfer, ISBN: 0-471-83093-3.

Rohsenow, W.M., Hartnett, J.P. and Cho, Y.I. (1998). Handbook of Heat Transfer, Third Edition, McGraw Hill.

Serna, M. and Jimenez, A. (2004). An Efficient Method for the Design of Shell and Tube Heat Exchangers, Heat Transfer Engineering, 25(2): 5–16.

TEMA (1999). Standards of TEMA, 8th Edition, Tubular Exchanger Manufacturers Association, New York.

DISEÑO Y SIMULACIÓN DE UN INTERCAMBIADOR DE CALOR TUBULAR VIRTUAL PARA APLICACIONES EDUCATIVAS

RESUMEN

Una de las tareas del diseñador de una planta industrial es la elección más adecuada de los aparatos que componen dicha planta, para que todos los dispositivos se complementen y estén sincronizados. Si en la planta industrial se disponen varios intercambiadores de calor, cada uno de ellos debe adecuarse a sus condiciones de funcionamiento particulares, las cuales serán función del proceso de trabajo que cada intercambiador realice. En el diseño y elección del intercambiador de calor más adecuado para un proceso de trabajo determinado, deberán respetarse unas normas internacionales y tenerse en cuenta las recomendaciones de las sociedades clasificadoras y los métodos utilizados en la industria actual. Con esto, teniendo en cuenta los métodos más avanzados en el diseño de intercambiadores de calor se crea un programa informático que nos permite diseñar un intercambiador de calor virtual adecuado a cada una de nuestras necesidades en el que podemos simular diferentes procesos térmicos y analizar su comportamiento en todo tipo de situaciones.

PROCESO DE DISEÑO DEL INTERCAMBIADOR DE CALOR

En el diseño de un intercambiador de calor para un proceso determinado se manejan una serie de datos y parámetros que lo van a caracterizar, siendo tarea del diseñador obtenerlos en función del proceso de trabajo para el que va a ser diseñado. Así mismo, el diseñador deberá elegir el método de diseño a seguir.

Definición del proceso

El cliente establece el proceso de transmisión de calor que necesita en su instalación, por ejemplo, un intercambiador de calor del tipo tubo-carcasa 1-2 en el que refrigerar 19 kg/s de un liquido orgánico de 71 ºC a 49 ºC utilizando como fluido refrigerante agua con una temperatura de entrada de 21 ºC, no debiendo exceder la temperatura de salida de 49 ºC, fijándose a continuación los siguientes aspectos:

— Material de construcción de los tubos: tubos de acero 14 SWG
— Diámetro exterior de los tubos: 19 mm
— Longitud de los tubos: 243.8 mm
— Disposición: triangular y 25.4 mm de paso entre tubos
— El factor de fouling por parte del fluido refrigerante: 0.00018 m^2 K/W

También se tendrán en cuenta las propiedades físicas del fluido a refrigerar en el intervalo de temperaturas de trabajo dadas:

— Viscosidad dinámica $\mu =$ 0.45 centiPoise
— Densidad $\rho =$ 882 kg/m^3
— Calor especifico cp = 0.93 KJ/kg K
— Número Prandtl $N_{pr} =$ 4.356

Las propiedades físicas del agua de refrigeración las calcula automáticamente el programa, por lo que no es necesario utilizar tablas ni con agua dulce ni con agua salada.

Crear un diseño nuevo

Se inicia el *módulo de diseño* en la ventana principal del programa (Figura 2) y se accede a la ventana de estimación primaria del intercambiador de calor introduciendo las siguientes variables en las ventanas correspondientes:

— Datos de identificación del intercambiador de calor
— Elección del tipo de intercambiador de calor
— Temperaturas del proceso
— Temperaturas medias calóricas
— Cálculo de la energía del proceso
— Dimensiones de los tubos
— Dimensiones de la carcasa
— Cálculo del Nº de Reynolds
— Coeficiente global de transmisión de calor
— Temperatura de la pared del tubo
— Coeficiente de convección en los tubos: en este módulo se calcula entre otros, el número de Nusselt, el coeficiente de convección y la caída de presión en el haz tubular.
— Cálculo de los parámetros del flujo y los deflectores en la carcasa
— Balance térmico y comprobación del diseño

PROGRAMA DE SIMULACIÓN

Descripción del programa

En la parte superior de la ventana principal del módulo de simulación y análisis del intercambiador de calor que se muestra en la Figura 4 se dispone un intercambiador de calor del tipo tubos-carcasa con las toberas de entrada y salida. Sobre el dibujo del intercambiador de calor hay sobrepuestas varias celdas que muestran datos sobre algunas de las características del proceso térmico y las dimensiones del intercambiador de calor. En las entradas del fluido refrigerado y del refrigerante se muestran la temperatura y el caudal y en las salidas se pueden ver la temperatura de los fluidos. Así mismo se presentan las dimensiones del intercambiador: longitud de los tubos, diámetro exterior e interior del tubo, paso entre los tubos y relación diámetro/paso tubular, número total de tubos, diámetro de la carcasa, disposición de los tubos en la placa tubular, espacio entre deflectores y número de pasos de los tubos y carcasa.

Para acceder a la información adicional de cada parámetro, basta con hacer doble clic sobre la celda para que aparezca un cuadro informativo más extenso.

En la parte inferior se muestra un cuadro de múltiples páginas sobrepuestas que nos ofrecen información sobre:

— Datos del proceso de intercambio de calor.
— Fouling.
— Gráficos.
— Análisis gráfico.

CONCLUSIONES

Se ha conseguido una herramienta rentable, fiable y didáctica para ayudar el ingeniero en el diseño de las plantas industriales y que permite comprender los procesos y mecanismos que tienen lugar en los intercambiadores de calor. Esta herramienta facilita las tareas del ingeniero permitiéndole diseñar fácilmente un intercambiador de calor, analizar y mejorar el intercambiador de calor diseñado, simular el funcionamiento del intercambiador de calor en diferentes condiciones de trabajo, simular y analizar el crecimiento del biofouling y sus efectos sobre el rendimiento del intercambiador de calor diseñado, así como analizar el Método de diseño recomendado por TEMA.

Los resultados de diseño obtenidos son comparables a los obtenidos en la práctica real en la industria, consiguiéndose un diseño óptimo que cumple con las funciones reales del intercambiador de calor.

La simulación de un intercambiador mediante ordenador es una herramienta complementaria y necesaria para obtener un diseño eficiente y económicamente rentable. Se puede estudiar el efecto de la variación de cualquier parámetro sobre los demás, sobre el comportamiento del intercambiador de calor y sobre su rendimiento. Además, las simulaciones que se pueden realizar son ilimitadas y dependen de las necesidades e imaginación del usuario.