HT2005-72760

A SIMPLIFIED CFD METHOD FOR THE DESIGN OF HEAT EXCHANGERS

D Brian Spalding
CHAM Limited
Bakery House
40 High Street
Wimbledon Village
London SW19 5AU
England

ABSTRACT

Traditional heat-exchanger design methods do not predict steady-state uniform-property performance well; and they are totally unable to predict the influences of time-dependence and varying properties or the consequent stresses in the shell and tubes.

On the other hand, conventional CFD (computational fluid dynamics) techniques, with their emphasis on body-fitting grids and sophisticated turbulence models, can contribute only to small-scale phenomena such as the velocity and temperature distributions within the space occupied by a few-tube sub-section of a tube bank.

Nevertheless, the practical importance of heat exchangers, including those which involve chemical reaction and phase change, is so great that engineers must find design tools which are both economically-affordable and more realistic in prediction than either of the just-mentioned extremes.

Such tools discretize space and time with the fineness allowed by modern computers; but they still inevitably employ space intervals which are large compared with tube diameters. They have been used for research purposes for many years; however, the difficulty of supplying them with all the relevant empirical input data has deterred designers from using them.

The lecture will describe a means of greatly reducing the difficulty; it accepts the formulae (for heat-transfer coefficients, viscosity-temperature relations, etc) in the form with which designers are familiar; and it also produces information, for example about local heat fluxes, hot-spots and stress concentrations, which otherwise escape attention.

Examples will be presented and explained.

Contents

1 THE HISTORICAL BACKGROUND

The first publication describing the application of CFD techniques for the simulation of heat exchangers appears to have been made more than thirty years ago by Patankar and Spalding [1] who concluded: "It therefore seems that a tool of considerable practical utility is in embryonic existence".

At first their expectations appeared to be fulfilled; for the same technique, generalized so as to be applicable to two-phase flows, played an important role in elucidating and resolving the practical difficulties which, in the mid-1970s, were being encountered by the nuclear-power industry.

Specifically, the shell-side steam-water mixture circulating in boilers, heated by pressurized water from the nuclear reactor, caused the tubes to vibrate and the baffles to corrode. Consequently, first Combustion Engineering Inc and Kraftwerk Union, then Babcock and Westinghouse, and finally the Electric Power Research Institute, sponsored the development of a family of flow-simulating computer programs.

The work was of a pioneering nature; and therefore did not proceed always as rapidly as desired. This prompted one wag to suggest that the name adopted for the EPRI-sponsored code, URSULA, was an acronym for Urgently Required Solution Unusually Late Arriving.

Despite the implied criticism, to which pioneers must become inured, the work was successful; and it was followed by the development of further computer codes for simulating steam condensers and cooling towers.

Nevertheless, the heat-exchanger-design community has not shown much enthusiasm for the use of CFD techniques; and the authors of a recent paper [2] on the subject concluded "very few applications can be found of using CFD technique as a tool for heat-exchanger design optimization". Instead, designers still prefer to use methods, for example those of Tinker [3] or Bell [4], in which the flow patterns are deduced from (educated) guesses rather than calculated.

The reasons for the failure of CFD techniques to attract the heat-exchanger-design community are not entirely clear. However, that they are in part psychological is suggested by the remarks of J Taborek [5] in the Hemisphere Handbook of Heat Exchanger Design. He there opines: "Only if calculations are performed manually will the engineer develop a 'feel' for the design process as compared to the impersonal 'black box' calculations of a computer program".

It is to be hoped that the approach recommended in the present paper will be found more congenial by heat-exchanger designers; for it enables them to insert the same formulae, including Tinker-Bell 'correction factors', which they would supply to the 'hand-held calculators' preferred by Taborek. Indeed, so great is the speed of advance of the computer-hardware and -software industries, that computers performing full CFD analyses may soon indeed be 'hand-held'.

The method to be described can be applied to heat exchangers of all types, to any participating fluids, and to any conditions of operation. However, in order to focus on essentials, discussion will henceforth be limited to:
- baffled shell-and-tube heat-exchangers,
- single-phase non-reacting fluids,
- steady-state operation, and
- thermal and pressure-drop performance.

2. THE REQUIREMENTS OF A HEAT-EXCHANGER DESIGN METHOD

2.1 Geometrical Input Data
No prediction is possible until the apparatus in question has been described in geometrical terms, which include (for the simplest cases):
- inside shell diameter
- inside shell-nozzle diameter
- tube outside diameter
- tube-wall thickness
- tube-layout pitch
- tube-layout characteristic angle
- tube length
- baffle cut
- baffle spacing
- number of tubes
- number of tube passes

2.2 Material Property Data
Specification must be made of:
- the thermal conductivity of the tube material
- the thermal conductivities of the shell- and tube-side fluids
- the specific heats of both fluids
- the densities of both fluids, and

- the viscosities of both fluids.

However, for most materials, these properties are known to vary with temperature; and this knowledge is expressed by way of:
- formulae,
- tables of numbers, or
- graphs of various kinds.

If graphs are in question, their content must be converted into formulae or tables before it can be communicated to a computer program. However, even when this has been done, the problem of using the information remains; for the whole point of a heat exchanger is to change temperature; and it is not known in advance what temperatures will prevail at any chosen point within the tubes or shell.

Therefore some means must be found of communicating to the computer program the whole content of the formulae or tables, together with the instruction: "You work out which values of conductivity and density etc to use at each point."

How this can be done is the main theme of the present paper.

2.3 Thermal and Mass-Flow Boundary Conditions
Also needed, of course, are the (known):
- mass-flow rate and temperature of the shell-side fluid in the inlet nozzle; and
- mass-flow rate and temperature of the tube-side fluid in its inlet header.

The task of performance prediction is to determine what will be the (mass-flow-weighted average) temperatures of the shell- and tube-side fluids at their outlets from the heat exchanger.

2.4 Empirical Correlations
If the geometry in question were extremely simple, as for example if there were only one tube and the shell had a length of many diameters and was free from baffles, and if the flow were laminar and of uniform temperature, it could be left to any well-constructed CFD program to work out the performance from the above data.

However, industrial heat exchangers have hundreds or thousands of tubes; and baffles are present and the flow is often turbulent. This entails that, if performance were to be predicted purely from computational fluid dynamics, a very fine grid would have to be employed. Even if a computer with sufficient memory could be found,

the time taken for the performance prediction would be orders of magnitude longer than any designer could afford to wait.

Moreover, so rudimentary is still the scientific knowledge of turbulence in flow patterns such as are found in tube banks, the reliability of the predictions would still be far from one hundred per cent.

The only practical solution is therefore to introduce additional information, derived from such experimental data as can be found, concerning the rates of heat and momentum transfer per unit area of solid-fluid interface. This information, which is the major outcome of thousands of man-years of heat-transfer and fluid-flow research, is usually expressed in the form of mathematically-expressed relationships between well-known 'dimensionless parameters':
- Nusselt or Stanton number, for the heat-transfer coefficient;
- Reynolds number, to characterize the state of the flow, and
- Prandtl number, to characterize the relative ease of heat and mass transfer in the fluid.

All these parameters involve the material properties listed in section 2.1; so the use of empirical correlations provides no escape from the need expressed there, namely to enable the computer program to work out the property values from the given formulae and the temperatures which it finds at every point.

2.5 Predicting the Flow Pattern and Temperature Distribution
The temperatures of the fluids leaving the heat exchanger are the main quantities which it is desired to predict; however, even if the flow pattern were as simple as that of the idealized one-dimensional counter-flow heat exchanger, these outlet temperatures depend on the temperature just upstream of the outlet. These just-upstream-of-outlet temperatures depend on the temperatures upstream of them; and so on. Therefore, the whole temperature distribution has to be computed.

When the flow pattern is not of the above simple kind, what point lies 'just upstream of' a given point is not obvious a priori; therefore ability to calculate the temperature distribution depends on ability to calculate the flow distribution giving rise to it. This therefore is what the computer program must additionally do, providing

incidentally two other pieces of information needed by the designer: the pressure losses suffered by the two streams.

Fortunately, computer programs (the so-called CFD codes) do exist for computing both the flow fields and the temperature distributions simultaneously. Although their accuracy depends on the fineness of computational grid which is employed, and desirably fine grids do increase computer times and therefore costs, the requirements relating to shell-and-tube heat exchangers are usually affordable.

However, just as the heat-transfer and friction correlations require material properties which vary with temperature according to formulae which must be made known to the code, they also contain other quantities which can not be specified *a priori*, namely the three components of the shell-side velocity.

It follows that, even if the temperature variations were small enough not to affect material properties, the need for the code to evaluate formulae from values which varied from place to place would remain. Thus the Reynolds Number enters most pressure-drop and convective-heat-transfer formulae; and its value depends on the local velocity, which varies with position in ways that are not known at the start.

In summary, predicting the performance of shell-and-tube heat exchangers necessitates use of a program with formula-processing capability.

3. THREE WAYS OF SATISFYING THE REQUIREMENTS

3.1 Method 1: 'User-Supplied Sub-Routines'
Of course, many CFD codes already have built-in correlation-evaluation sequences, representing friction and heat-transfer processes; and they also contain computer-coding modules which express the variations with temperature and pressure of the relevant properties of frequently-encountered materials.

In principle, there is no limit to the extent to which these provisions can be extended. But in practice, however much is provided, some users of the code will require more; they will want it at once; and they will not want to pay the costs incurred by the code-developer in providing it.

From the earliest years of commercial CFD, therefore, developers have allowed users to add

coding modules of their own, usually in the form of Fortran or C subroutines, which would supplement the built-in correlations in the desired direction.

Users of the 1981 PHOENICS code, for example, will remember what clever use some users made of the so-called 'GROUND-coding' facility, which indeed many old-stagers continue to use. Reference [1] is an excellent example of the use of this technique.

However, the proportion of CFD-code users with the necessary skills is constantly diminishing; and the proportion of heat-exchanger designers who possess (or have the time to acquire) them must be very small.

3.2 Method 2: 'Automated Sub-Routine Writing'
In order to enable PHOENICS users to benefit from the features of 'GROUND-coding' without themselves having to be familiar with either Fortran or C, the so-called 'PLANT' feature was introduced in 1997.

This enabled the user to express his wishes by way of formulae, written in accordance with prescribed rules; whereupon PHOENICS itself:
- interpreted the formulae;
- created corresponding Fortran subroutines;
- compiled them;
- re-built the executable; and
- carried out the required flow-simulating calculation.

This was a big step forward; and it did, at least potentially, satisfy the 'formula-processing' requirement which has been pointed out above. However, perhaps because it was not adequately presented to them, it did not convert many heat-exchanger designers into CFD users. Perhaps also the 'prescribed rules' were shaped by those thinking too much of the Fortran to be written, and not sufficiently of the prospective user.

PLANT remains as a valuable feature of the current PHOENICS; but the feature now to be described has rendered it almost redundant.

3.3 Method 3: When the Program Understands Formulae
The new feature has the acronym "In-Form", standing for "Input (or Intake) of Formulae". Its concept is very simple, namely:

- let the user write, in language which he or she can understand easily, the formulae according to which the solver is to compute values of, say, heat-transfer coefficient;
- let this be sent to the equation-solving module in the form of a character string;
- let the latter module first inspect and parse all the strings which it receives, converting them into settings of 'internal switches' which dictate how the calculation shall be conducted; and then
- let the calculation proceed.

The formulae relevant to heat-exchanger design are those which:
- compute the local material properties in shell-side fluid, tube-side fluid and metal, from the local temperatures and (if appropriate) pressure;
- compute the corresponding Reynolds and Prandtl numbers;
- compute the corresponding Nusselt or Stanton numbers and thence the local heat-transfer and friction factors;
- compute the resulting local sources and sinks of heat and momentum, and
- supply these to the equations on which the solver operates.

In-Form has many more capabilities than the heat-exchanger designer is likely to need. Therefore only the few relevant ones will be illustrated here, as follows:
- specific fluids may be introduced by statements such as:
  "tube_fluid is SAE_5W-30_engine_oil",
  which dictates use of a pre-existing set of formulae for density, viscosity, specific heat and thermal conductivity;
- individual material properties can be set by such statements as:
  "property RHO1 at SHELL is CONST1+CONST2*TEM1+CONST3*TEM1^2",
  where ^ is the symbol used for exponentiation;
- quantities such as local Reynolds number can be set by statements such as:
  "REYN is TUBEDIAM*RHO1*ABSV/VISC";
  then Stanton Numbers can be specified by the statement
  "Stored var STAN is CONST1*REYN^CONST2*PRNO^CONST3"
  whereafter a local heat-transfer coefficient can be specified by way of
  "Stored var COEF is STAN*RHO1*CP1*ABSV";

and finally
- the heat source can be specified via:
  "Source of TEM1 is COEF*AREA*(TMET-TEM1)".

The relevance of these statements to heat-exchanger design should be obvious; for they say (nearly) all that is to be said about how local heat transfer from tube to fluid is to be calculated. Once they have been typed into the input file, there is nothing further for the designer to do than to wait for the calculations to be completed.

4. PRACTICAL EXAMPLES

4.1 The Heat-Exchanger and Computer Code in Question

A heat exchanger will be considered which has a single tube pass and a shell of 4m length and 1m diameter with three baffles. The shell-side fluid is water and the tube-side fluid SAE 5W-30 engine-oil; these enter at 10 and 60 degrees Celsius respectively. The mass flow rates of both fluids are 100 kg/s.

Text-book formula, connecting Nusselt, Reynolds and Prandtl numbers, have been adopted for the shell-side and tube-side heat transfer coefficients and the distributions of pressure, velocity and temperature of the two fluids have been computed by the general-purpose CFD code, PHOENICS, without activation of any heat-exchanger-specific special features. Therefore any other CFD code equipped with a formula-understanding module could have been used.

The computer runs of which the results will be presented have been chosen in order to illustrate:
- that the variations in space of properties, flow patterns, Reynolds numbers, Prandtl numbers, Nusselt numbers and heat-transfer coefficients, which are neglected in conventional design methods, are rather large;
- that it is however as easy to calculate, and to take account of, these variations as it is to neglect them; and
- that changing the formulae used, the output required or any other specification, and then observing the consequences, is a matter of minutes only.

The distributions are of course three-dimensional. However it is sufficient for present purposes to consider the central plane only.

Results will be represented by way of vector and contour plots on this plane.

After presentation of the results, some of the formulae used for generating them will be shown and discussed.

4.2 Distributions of Velocity, Temperature, Pressure and Related Properties

The velocity distribution in the shell, on the central plane is shown in Fig. 1. The shell-fluid inlet is at the bottom on the left, and its outlet at the top on the right. The influence of the two baffles is clearly seen.

The temperature distributions in the shell- and tube-side fluids are shown in Figs. 2 and 3. The latter conforms to that to be expected in an ideal counter-flow heat-exchanger, but the former (understandably) does not.

The pressure distribution in the shell is shown in Fig. 4. It shows the to-be-expected discontinuities caused by the baffles.

As has been emphasized above, the properties of most fluids depend upon temperature. The variation of the viscosity of the tube-side fluid, for example, is shown in Fig 5. Evidently (see scale on the right), it varies by a factor of more than two,

4.3 Distributions of Reynolds, Prandtl and Nusselt Number

The dimensionless parameters which feature in the calculation of heat-transfer coefficients vary through the heat exchanger, partly because of temperature variations and partly because of the non-uniformities of velocity. The distributions of Reynolds, Prandtl and Nusselt number calculated in the present case are shown, for the shell- and tube-side fluids in turn, in Figs. 6 to 11.

In respect of Reynolds number, variations are greatest on the shell side; the presence of the baffles accounts for this. The consequential variation of Nusselt number is correspondingly great, being highest near the edges of the baffles and fifty times smaller in low-velocity regions near the baffle roots and in the corners opposite inlet and outlet.

4.4 Variations of Heat-Transfer Coefficient

From the Nusselt numbers, the computation of the corresponding shell- and tube-side heat-transfer coefficients per unit volume of shell is an easy step; and the overall heat-transfer coefficient follows as their harmonic mean. Their distributions are displayed in Figs. 12, 13 and 14.

Evidently the shell-side coefficients show the widest variations but, their effect on the all-important <u>overall</u> coefficient is diminished by the fact that the tube-side coefficients are the smaller. The maximum value is therefore less than 50% greater than the minimum value.

This last observation might be regarded as countering, to some extent, the criticism of conventional design methods which has been implied above; but it does not truly do so. The objection to conventional method is not that that they use a <u>constant</u> value of 'U' for the whole heat exchanger; it is that the means of determining that value are based on guesswork and <u>not on detailed analysis of the flow field.</u>

4.5 How the Input Data Were Introduced

The results displayed so far will have come as no surprise to any heat-exchanger designer who has thought even superficially about what is truly happening inside his equipment. What may interest him more may be to learn:
(a) how easily they were obtained, and
(b) how quickly the effects of changes to the input can be investigated.

The ability of the computer program to understand easily written formulae is the key requirement. Examples of such formulae will now be presented, starting with the following extract from the engineer's data-input file:

(stored var REY1 is DIAM*VABS/ENUL)
(stored var PRN1 is CPS*RHO1*ENUL/COND)
(stored var NUS1 is 0.2*REY1^0.6*PRN1^0.33)
(stored var COE1 is
AOVERV*NUS1*COND/DIAM)

(stored var REY2 is DIAM*TUBVEL/ENU2)
(stored var PRN2 is CPT*RHO2*ENU2/CON2)
(stored var NUS2 is
MAX(2.0,0.328*(REY2*PRN2)^0.33))
(stored var COE2 is
AOVERV*NUS2*CON2/DIAM)

(stored var COEU is 1/(1/COE1+1/COE2))
(stored var TEMM is
(COE1*TEM2+COE2*TEM1)/(COE1+COE2))

The lower-case words ('stored', 'var', 'is') tell the code that it must compute some new variables everywhere within the 'virtual heat exchanger' with which it is concerned. The upper-

case words have been chosen by the engineer, who appears to have found:

- REY1 an adequate name for 'Reynolds number of shell-side fluid',
- REY2 likewise for 'Reynolds number of tube-side fluid',
- PRN1, PRN2, NUS1 and NUS2 as adequate for their Prandtl and Nusselt numbers,
- that AOVERV will serve as 'area/volume',
- COE1, COE2 and COEU remind him of the shell-side, tube-side and overall heat-transfer coefficients,
- and so on.

The engineer has also chosen the formulae, most of which are rather easy to interpret. Thus:

- 'DIAM*TUBVEL/ENUL stands for 'diameter times tube-fluid velocity divided by viscosity, i.e. the tube-side Reynolds number, wherein the first two variables (the code is clever enough to work out) are constants, whereas the third must be computed for each location;
- 'NUS2 is
  MAX(2.0,0.328*(REY2*PRN2)^0.33)'
  represents a formula which some text books recommend for pipe flow at modest Reynolds numbers, with a lower limit of 2.0 inserted by the engineer for some reason of his own;
- '1/(1/COE1+1/COE2))' shows how the overall coefficient is to be computed from the individual coefficients; and
- the formula for the tube-metal temperature, TEMM, shows that it is a suitably-weighted mean of the fluid temperatures, all varying, like the coefficients themselves, from place to place.

How were the material properties calculated? Also by way of formulae drawn from textbooks. Thus, the formulae for the properties of the engine-oil in question were found in a book by K. Hagen [6] and translated thus into the form understood by (at least) the PHOENICS computer code:

(property rho2 is 1052.3-0.6420*TEM2)
(stored var cpt is 753.7+3.65*TEM2
(stored var con2 is 0.1447-2.3073E-5*TEM2)
(stored var enu2 is 10.0^(POL4(TEM2,58.2987,-.53817,1.92827e-3, -3.16448E-6,1.97922E-9)-2)

wherein POL4(TEM2,…..) signifies a fourth-power polynomial, the coefficients in which are the numbers which follow the comma. Fortunately, many such formulae are provided as items to be

selected from a library. Nevertheless, it is not difficult to translate <u>any</u> formula into 'In-Form-speak': users are not confined to what is on the code-vendors' menu.

4.6 Discussion
There are engineers who are unwilling to do more, when designing equipment, than make selections, by way of mouse-clicks, from what so many have done before that the code provider has had time and inducement to adopt as built-in options. Others however, more like those referred to in the above quote from J.Taborek, will welcome the power which In-Form provides to "develop a 'feel' for the design process".

Such an engineer, noting the considerable temperature differences between the metal temperature and the tube-fluid temperature, might wonder whether he ought to multiply the relevant coefficient, as some textbooks recommend, by the ratio (wall viscosity / bulk viscosity) raised to the power 0.14). He could set his doubts at rest in a few moments by introducing the following lines into his input file, by copying:

(stored var ENUM is:
10.^(POL4(:TEMM,58.2987,-.53817,1.92827e-3, -3.16448,E-6,1.97922E-9)-2)

so as to compute the viscosity of the engine-oil at the metal-wall temperature, TEMM; then he should modify the formula for COE2 so as to reflect the viscosity-ratio effect, thus:

(stored var COE2 is
(ENUM/ENU2)^0.14*AOVERV*NUS2*CON2/DIAM).

If he did this he would learn, after inspecting the computed results, that the total heat transferred would increase by only 0.5 %, so adding to adding to the stock of experience which engineers call 'feel'.

Alternatively, he might wish to know how the heat-transfer rate per unit volume varied through the shell; then he could satisfy his curiosity by writing the single line:

(stored var FLUX is COEU*(TEM2-TEM))

and, one minute, later view the results presented in Fig. 15. Comparing this with other images which he has seen before he will be able to decide whether he is satisfied with the nearly-two-fold variation there disclosed, or will seek by re-design to achieve greater uniformity.

Lastly, he might suddenly think: "What about fouling?" Then he has only to insert one line in his input file such as:

(stored var FOUL is …. *whatever function of position and temperature he or she invents*)
and to change the overall-coefficient line to:

(stored var COEU is 1/(1/COE1+1/COE2)+FOUL)

The subsequent computer run will inform him of the consequences, both by way of numbers and of graphical displays.

5. CONCLUDING REMARKS
There is no limit to the easily-effected variations of input and output which the In-Form facility allows; so, the main points having already been made, no further examples will be presented now.

It should however be stressed that the techniques employed can be applied also when:
- the heat exchanger is of any other type, whether or not specified by TEMA;
- the heat-exchanger conditions vary with time (e.g. when the inlet mass-flow rates and/or temperatures vary in accordance with piece-wise-linear formulae);
- either or both fluids are two- or multi-phase, In-Form being used to introduce any of the complex formulae which are commonly used for the design of such equipment [7];
- chemical reaction takes place within the tubes in accordance with a formula which depends not only on local temperature but also on the changes in composition to which the reaction itself gives rise;
- it is desired also to compute the stresses which are present in the load-bearing elements of the equipment, which stresses of course vary with position and time in accordance with the calculated distributions of temperature and pressure.

In summary, it has been argued that the time has come for heat-exchanger designers to exploit the now-well-established techniques of computational fluid dynamics. Specifically, the obstacles which have prevented their doing so, namely the difficulty of introducing the necessary empirical knowledge, have been removed by the recently-developed In-Form module. This enables the computer code to 'understand', and incorporate into its calculations, the formulae which designers are accustomed to use in their traditional methods.

Thus all that formerly could be done only by specialists capable of exploiting the 'user-subroutine' facility offered by some CFD-code vendors, can now be done by any flexibly-minded heat-transfer engineer.

6. ACKNOWLEDGMENTS

7. REFERENCES

[1] Patankar S V and Spalding D B, 1974, "A calculation procedure for the transient and steady behavior of shell and tube heat exchangers" in *Heat Exchangers: Design and Theory Sourcebook* ed. Afgan A and Schluender E, McGraw Hill.
[2] Stevanovic Z, Ilic G, Radojkovic N, Vukic M, Stefanovic V and Vuckovic G, 2001, "Design of shell and tube heat exchangers by using CFD technique – part one: thermo-hydraulic calculation". Facta Universitatis Series Mechanical Engineering vol 1 no 8, pp1091 - 1105.
[3] Tinker T, 1958, *J. Heat Transfer* vol 80 pp 36-52.
[4] Bell K J, 1993, "Final report of the cooperative research program on shell-and-tube heat exchangers" University of Delaware *Exp.Sta.Bull. 5.*
[5] Taborek J, 1983, "Recommended method: principles and limitations" in *Hemisphere Handbook of Heat Exchanger Design* ed. Hewitt G F, Hemisphere, New York.
[6] Hagen K, 1999, "Heat transfer, with applications", Prentice Hall,
[7] Chisholm D, 1988, "Two-phase flow in shell-and-tube heatexchangers" in "Heat Exchanger Technology" ed. Chisholm D, Elsevier.

8 FIGURES

Figure.1 Velocity vectors on the
shell side of the heat exchanger

Figure 2. Shell-side temperature contours

Figure 3 . Tube-side temperature contours

Figure 4. Pressure distribution in the shell

Figure 5 . Tube-side-fluid viscosity distribution

Figure 6. Shell-side Reynolds-Number contours

Figure 7. Shell-side Prandtl-Number contours

Figure 8. Shell-side Nusselt-Number variation

Figure 9. Tube-side Reynolds-Number contours

PRN2
176
192
208
224
240
255
271
287
303
319
334
350
366
382
398

Figure 10 . Tube-side Prandtl-Number contours

NUS2
5.6
5.6
5.6
5.7
5.7
5.7
5.7
5.7
5.7
5.7
5.7
5.7
5.7
5.7
5.7

Figure 11. Tube-side Nusselt-Number contours

COE1
7.33E+4
1.73E+5
2.73E+5
3.73E+5
4.73E+5
5.73E+5
6.73E+5
7.72E+5
8.72E+5
9.72E+5
1.07E+6
1.17E+6
1.27E+6
1.37E+6
1.47E+6

Figure 12. Shell-side heat-transfer coefficient contours

Figure 13 . Tube-side heat-transfer-coefficient contours

Figure 14 . Overall-heat-transfer-coefficient contours

Figure 15 . Heat flux contours