The current issue and full text archive of this journal is available at
www.emeraldinsight.com/0961-5539.htm

Simulation of compact heat exchanger performance

Bengt Sunden
Department of Energy Sciences, Heat Transfer, Lund University, Lund, Sweden

Received 30 January 2009
Accepted 2 February 2010

Abstract

Purpose – The purpose of this paper is to present some methods to analyse and determine the performance of compact heat exchangers; show the applicability of various computational approaches and their limitations, provide examples to demonstrate the methods, and present results to highlight the opportunities and limitations of the considered methods.

Design/methodology/approach – Engineering methods based on thermal balances and correlations, as well as computational fluid dynamics (CFD) methods based on the finite control volume (CV) approach, are used.

Findings – Overall, it is found that computational heat transfer methods of various kind and complexity are useful tools if carefully handled and appropriately applied. However, there are several constraints, difficulties and limitations to be aware of. Radiators, extended surfaces and enhanced ducts are considered.

Originality/value – The paper presents a timely and coherent review and description of various computational methods to simulate the thermal-hydraulic performance of compact heat exchanger issues.

Keywords Heat exchangers, Heat transfer, Correlation analysis, Simulation

Paper type General review

Nomenclature

A	area
B	thickness
C	heat capacity flow rate
D	diameter, diffusion flux
F	friction factor
F	correction factor LMTD
G	mass velocity
K	coefficient in pressure drop expression
L	characteristic length
NTU	number of transfer units
Nu	Nusselt number
$\dot{Q}$	heat transfer rate
S	surface, source term
U_j	velocity vector
u_j	velocity vector
V	volume, average velocity
x_i	coordinate

Greek symbols

α	heat transfer coefficient
Δp	pressure drop
ε	efficiency
ϕ	arbitrary dependent variable
Γ	general diffusivity
λ	thermal conductivity
ρ	density
σ	area ratio

Subscripts

c	contraction
D	Darcy
e	expansion

Financial support for several research projects related to this paper has been received from the Swedish Energy Agency (STEM), the Scientific Council (VR) and some companies. Among these are Alfa Laval AB, Nilcon Engineering AB and Valeo Engine Cooling Systems.

International Journal of Numerical Methods for Heat & Fluid Flow
Vol. 20 No. 5, 2010
pp. 551-569

0961-5539
DOI 10.1108/09615531011048240

f	face	m	mean
front	front face	min	minimum
F_i	fouling inside	max	maximum
F_o	fouling outside	out	out
h	hydraulic	o	outside
in	in	w	wall
i	inside	vl	heat conduction

1. Introduction

Compact heat exchangers for gas flows are characterized by very high heat transfer area per unit volume, typically above 7,000 m^2/m^3 with hydraulic diameter of the flow passages less than 6 mm. There is now development towards mini and microchannel flow passages with significant improvement in the heat transfer area surface density like so-called mesoscale heat exchangers having about 3,000 m^2/m^3 with hydraulic diameter less than 1 mm and microscale heat exchangers having about 15,000 m^2/m^3 with hydraulic diameters below 100 μm. Liquid and phase change heat exchangers are regarded compact if the heat transfer surface area density is greater than about 400 m^2/m^3. Due to the current high energy costs, concern for environmental protection and request for sustainable development a renewed interest has appeared for high performance heat exchangers and not the least compact ones. In the design and development processes, one needs to calculate or estimate the performance of compact heat exchangers and for this some different methods are possible. One method is to adopt an engineering approach based on overall data and such approaches are generally described in textbooks or handbooks in heat transfer (Shah and Sekulic, 2003; Sunden, 2007a; Incropera nd and DeWitt, 1996). Conveniently rating and sizing issues can be handled. Another method is to adopt so-called computational fluid dynamics approaches (CFD) which also allow detailed analysis of basic transport processes within a heat exchanger. CFD is particularly useful in the initial design steps reducing the number of testing of prototypes and providing a good insight in the transport phenomena occurring in the heat exchanger. Also heat exchanger optimization is important for process intensification and here also CFD can be a helpful tool. Failure investigation is another opportunity. For severe situations where very high pressure prevails, very hot or cold or toxic fluids are involved, quick results are needed, accurate CFD tools would be ideal. The basic principles of CFD might be found in Versteeg and Malalasekera (2007), Ferziger and Peric (1996). This paper describes the mentioned computational procedures for compact heat exchangers. Constraints, problems, difficulties and limitations are outlined. Some results are shown. What approach being recommended depends on the overall purpose of the calculations, i.e. related to what kind of results being of interest.

1.1 Examples of compact heat exchangers and flow passages

Figure 1 presents a few compact heat exchangers as well as an example of flow passages.

1.2 General research needs

There are several existing and potential application areas of compact heat exchangers, e.g. automotive, electronics cooling, fuel cells, air conditioning and refrigeration, aerospace, gas turbines, renewable energy and other industrial applications. For

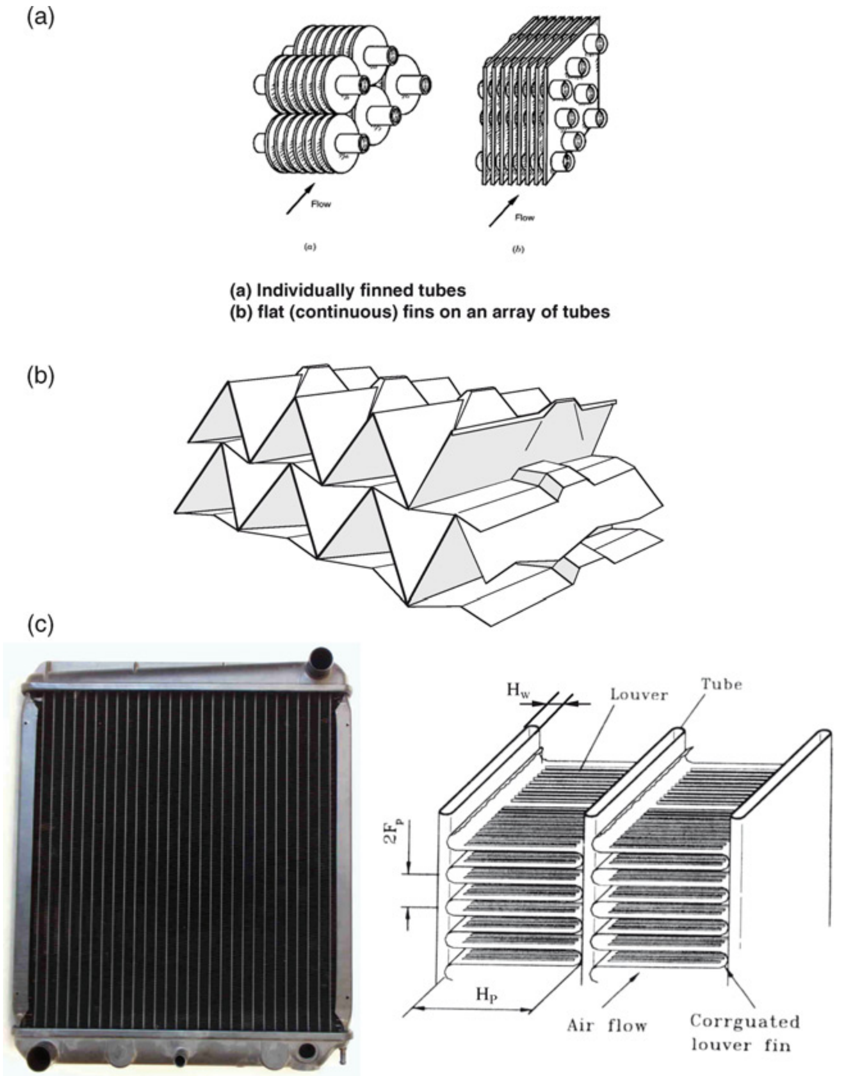


Figure 1.

Notes: (a) Simple compact heat exchangers; (b) triangular flow passages with bumps in advanced regenerators; and (c) vehicle radiator (air-liquid heat exchanger)

promotion and introduction of such equipment, the underlying mechanisms of heat transfer and fluid flow transport phenomena need to be addressed. Research needs are requested for achieving the following goals:

- how to achieve more compact heat exchangers;
- high thermal efficiency;
- balance between enhanced heat transfer and accompanied pressure drop;
- mini and micro channel heat transfer and pressure drop characteristics;

- material issues especially for high-temperature applications;
- manufacturing methodology;
- fouling; and
- non-steady operation.

2. Engineering approaches

2.1 The logarithmic mean temperature difference method for analysis of heat exchangers

Figure 2 shows the principle of the heat transfer from the hot fluid to the cold fluid.

In the overall thermal analysis of heat exchangers, the total heat transfer rate Q is of primary interest. To get started, the overall heat transfer coefficient U is assumed to be constant in the whole heat exchanger (average value). The heat transfer rate is then written:

$$\dot{Q} = UA \cdot F \cdot LMTD \tag{1}$$

where A is the heat transferring area and $LMTD$ is the logarithmic mean temperature difference between the hot and cold fluids if they were flowing in counter flow arrangement. F is a correction factor. For heat exchangers not operating in counter flow, the correction factor F is in the range of $0 < \mathrm{F} \leq 1$. F depends on the heat exchanger type and two additional parameters representing the ratio of heat capacity flow rates and an efficiency or goodness number, respectively. The overall heat transfer coefficient U is found from equation (2) below:

$$\frac{1}{UA} = \frac{1}{\alpha_{\mathrm{i}} A_{\mathrm{i}}} + \frac{1}{\alpha_{\mathrm{Fi}} A_{\mathrm{i}}} + \frac{b_{\mathrm{w}}}{\lambda_{\mathrm{w}} A_{\mathrm{vl}}} + \frac{1}{\alpha_{\mathrm{F_o}} A_{\mathrm{o}}} + \frac{1}{\alpha_{\acute{o}} A_{\mathrm{o}}} \tag{2}$$

where α_{i} is the heat transfer coefficient on the inside, A_{i} the convective heat transfer area on the inside, $\alpha_{\mathrm{F_i}}$ the fouling factor on the inside, b_{w} the thickness of the intermediate solid wall, λ_{w} the thermal conductivity of the wall material, A_{vl} the heat conducting area, $\alpha_{\mathrm{F_o}}$ the fouling factor on the outer surface, α_{o} is the heat transfer coefficient on the outer surface and A_{o} the convective heat transfer area on the outer side. The resistance due to heat conduction in equation (2) is only valid for a plane wall or if the material thickness is very small.

The heat transfer coefficients α_{i} andα_{o} are determined as described in section 2.3

2.2 The ε – number of transfer units method for analysis of heat exchangers

In the analysis of the performance of a certain heat exchanger the amount of heat being transferred, outlet temperatures of the fluids and the pressure drops are of most interest. At the design and sizing stage of a heat exchanger, the heat transferring area

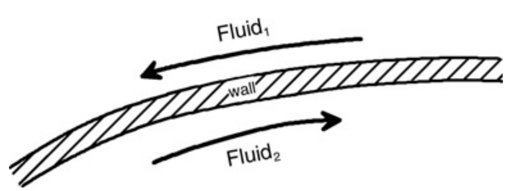


Figure 2.
Convective-conductive-convective heat transfer from the hot to the cold fluid

and other dimensions are determined in such a way that a prescribed heat flow can be transferred and that the pressure drops are within permitted limits.

If the inlet and outlet temperatures of the hot and cold fluids are given, the *LMTD* – method is quite suitable. In other cases, the so-called $\varepsilon - NTU$ method is more appropriate.

The efficiency or effectiveness ε is defined as the ratio between the real heat transfer rate and the maximum possible heat transfer rate:

$$\varepsilon = \frac{\text{real amount of heat}}{\text{maximum possible transferrable amount of heat}} = \frac{\dot{Q}}{\dot{Q}_{\max}} \tag{3}$$

$$NTU = \frac{UA}{C_{\min}}. \tag{4}$$

Relations between ε and NTU are available for some heat exchanger configurations in textbooks, e.g. (Shah and Sekulic, 2003; Sunden, 2007a; Incropera nd and DeWitt, 1996). ε is a function of C_{min}/C_{max} (the heat capacity flow rate ratio), NTU and heat exchanger type.

2.3 Correlations for the heat transfer coefficients

In the design or sizing of a heat exchanger, the heat transfer coefficients on the inner and outer walls of the tube or duct must be calculated. Summaries of the various correlations for convective heat transfer coefficients for internal and external flows can be found in Shah and Sekulic (2003), Sunden (2007a), Incropera nd and DeWitt (1996), commonly expressed in non-dimensional Nusselt numbers. In addition, the friction coefficient is needed for determination of the pumping power requirement. The Nusselt number is generally expressed as:

$$\begin{aligned} \mathrm{Nu} &= \alpha L/\lambda \\ &= \text{function (flow velocity, physical properties, geometry)} \\ &= \text{function (Re, Pr, geometry)}. \end{aligned} \tag{5}$$

Such correlations can be established by experimental investigations, analytical or semi-empirical computational methods and CFD procedures.

2.4 Pressure drop in heat exchangers

The pressure drop on the gas side in compact heat exchangers is usually splitted up in three components, namely the frictional loss, acceleration of the fluid and inlet and outlet losses. For tubular heat exchangers with plane fins as in Figure 1 and if the gas is passing the tubes in cross flow, the pressure drop is calculated as:

$$\Delta p = \frac{G^2}{2\rho_{\mathrm{in}}}\left[(1+\sigma^2)(\rho_{\mathrm{in}}/\rho_{\mathrm{out}} - 1) + f\frac{A}{A_{\min}}\frac{\rho_{\mathrm{in}}}{\rho_{\mathrm{m}}}\right]. \tag{6}$$

In equation (6) ρ_{in} is the density at the inlet, ρ_{out} is the density at the outlet and G is the mass velocity. The area ratio σ is determined according to:

$$\sigma = \frac{A_{\min}}{A_{\mathrm{front}}}. \tag{7}$$

The average density ρ_m is calculated according to:

$$\frac{1}{\rho_m} = \frac{1}{2}\left(\frac{1}{\rho_{in}} + \frac{1}{\rho_{out}}\right). \tag{8}$$

For finned plate heat exchangers, see Figure 1, the pressure drop is calculated according to:

$$\Delta p = \frac{G^2}{2\rho_{in}}\begin{bmatrix} (K_c + 1 - \sigma^2) + 2(\rho_{in}/\rho_{out} - 1) + f\dfrac{A}{A_{min}}\dfrac{\rho_{in}}{\rho_m} - (1 - K_e - \sigma^2)\dfrac{\rho_{in}}{\rho_{out}} \\ \text{inlet} \qquad\qquad \text{acceleration} \qquad\qquad \text{friction} \qquad\qquad \text{exit} \end{bmatrix} \tag{9}$$

In equation (9), K_c is a contraction coefficient at the inlet and K_e is an expansion coefficient at the outlet. Typical values of K_c and K_e are given in Kaysand and London (1984).

In the description of calculation methods for the pressure drop in compact heat exchangers and shell-and-tube heat exchangers the frictional pressure drop appears.

The frictional pressure drop inside a heat exchanger results when fluid particles move at different velocities because of the presence of structural walls such as tubes, shell, channels, etc. It is calculated from a well-known expression as:

$$(\Delta p)_{friction} = f_D \frac{L}{D_h}\left(\frac{1}{2}\rho V^2\right) \tag{10}$$

where f_D is the Darcy friction factor = 4f and f is the Fanning friction factor; D_h is the hydraulic or equivalent diameter = 4 (flow area)/(flow perimeter); and $\rho V^2/2$ is the dynamic pressure. The Darcy friction factor, f_D, is available for a variety of simple basic geometries and flow conditions in textbooks, e.g. (White, 2003).

2.4.1 Pressure drop from area change. Pressure drop from area change occurs as a result of energy dissipation associated with eddies formed when a flow area is suddenly expanded or contracted. It is expressed in the following form:

$$(\Delta p)_{area\ change} = K\left(\frac{1}{2}\rho V_{max}^2\right), \tag{11}$$

where K is the pressure loss coefficient and V_{max} is the flow velocity based on smaller or minimum flow area. K is a function of flow-area ratio and is available in textbooks or handbooks. Equation (11) may be used to calculate pressure drops from inlet nozzles to shell or channel, from shell or channel to outlet nozzles, and tube side inlet and exit losses.

2.4.2 Pressure drop owing to flow turning. When a fluid turns along a curved surface or mitered bend, a secondary flow is formed as a result of centrifugal force acting on fluid particles. An energy dissipation follows, and the pressure decreases. The pressure drop associated with flow turning is expressed as:

$$(\Delta p)_{turning} = K\left(\frac{1}{2}\rho V^2\right) = f_D\left(\frac{L}{D_h}\right)_{equivalent}\left(\frac{1}{2}\rho V^2\right) \tag{12}$$

where K is the turning-loss coefficient, V is the flow velocity calculated based on the upstream unaffected flow area, and L/D_h is an equivalent length-to-diameter ratio. The equivalent length-to-diameter ratio concept can also be used in conjunction with a familiar friction pressure-drop formula of equation. The pressure drops in a U-bend section of U-tubes in a heat exchanger can be calculated using a similar approach as given in equation (12).

2.5 Balancing heat exchanger pressure drop vs fan and pump characteristics

To have the heat exchanger working properly fans and pumps need to be matched properly with the heat exchanger characteristics. In this section the balancing issue of the airside of a radiator is considered. From the formulas given above the pressure drop vs volume flow rate for the heat exchanger can be established. However, as several not coinciding correlations are available one needs to be careful in interpreting the results and selection of fan. Figure 3 shows the pressure drop on the airside by using three different correlations for the friction factor. As is evident the deviation is large. The fan characteristic (at a certain rotation speed) is also included. The intersection between the fan and heat exchanger curves gives the corresponding operating point. It is obvious that at a certain pressure drop the fan will give different flow rates. Thus, it is clear that accurate correlations are needed to design the system properly. Also, one needs to avoid the unstable operating regime of the fan.

3. Computational fluid dynamics – heat transfer and fluid flow analysis

As the flow pattern is important for the convective heat transfer process both the flow and thermal fields have to be considered as research and development for improvement are attempted. In heat exchanger applications many duct configurations are employed. Enhancement of heat transfer on one or both sides of the heat exchanger is attempted for to reduce the size of the equipment for a given duty, increase the capacity of an existing apparatus, or decrease the approach temperature difference. Common enhancement techniques are offset strip fins, wavy fins, tube inserts and vortex generators. Also the duct walls might be corrugated or wavy. As measurement procedures become quite difficult and it is a formidable task to obtain whole field information particularly for complex and narrow geometries it would be ideal if some form of computer simulations could be carried out. CFD is nowadays an established name for a technique to model fluid flow and heat transfer (and other transport phenomena as well) using computer simulations. The rapid growth of powerful

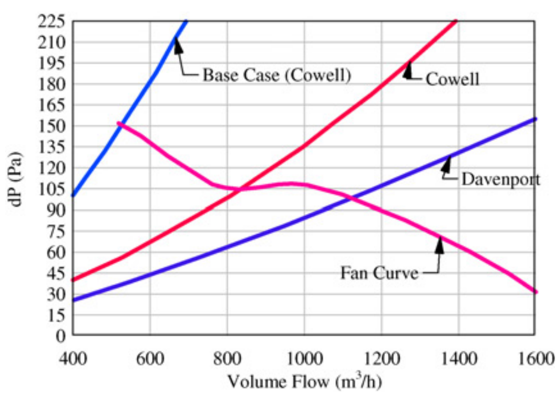


Figure 3.
Heat exchanger pressure drop characteristics on the airside based on three different correlations and the fan characteristic operation curve

computer resources and the development of general purpose CFD software packages have created a tool for simulation of industrial thermal problems. Nevertheless, several shortcomings still exist but progress is continuously achieved. The CFD tool can be applied to simulate the flow and temperature fields, the pressure drop and heat duty. The temperature fields can be used to calculate heat transfer coefficients which can be applicable in structural analysis of the equipment. Then some evaluation vs the design specifications can be made. If needed an iterative process can be performed and the geometry data can be changed and the simulations be repeated until satisfactory results have been achieved. The fluid-structure coupling (becoming more and more important) is of a sequential type allowing various codes to be applied. However, recently development of CFD methods to directly couple the solid-fluid and thermal analyses has been promoted (Spalding, 2006). Nowadays, computational tools are able to handle geometries of almost any arbitrary shape.

The geometry has to be represented by a computational mesh consisting of a large number of cells or control volumes (CVs). All equations are solved in all these CVs according to the conservation equations describing the transport phenomena (momentum, heat, mass). Special mesh generating software produces meshes based on computer-aided-design (CAD)-data representing the three-dimensional geometry. Such mesh generators enable convenient adaptation of the mesh to modifications in the CAD-data. The accuracy and resolution of the results obtained depend on the number and distribution of the CVs, convection-diffusion schemes being applied and how the turbulence is modelled. The results from the simulations can be visualized as vector plots for flow velocities, contour plots for scalar quantities like pressure, temperature etc. Many CFD packages have visualization tools and can also be linked to such independent tools. Animations are also possible and such opportunities are particularly useful for unsteady or transient problems.

CFD can be applied to heat exchangers in different ways. One way is to model the entire heat exchanger or the heat transferring surface. This can be done by using large scale or relatively coarse computational meshes or by applying a local averaging or porous medium approach. For the latter case volume porosities, surface permeabilities as well as flow and thermal resistances have to be introduced. The porous medium approach was first introduced by Patankar and Spalding (1974) for shell-and-tube heat exchangers and has later been followed up by many others and not only for shell-and-tube heat exchangers.

Another way is to identify modules or group of modules which repeat themselves in a periodic or cyclic manner in the main flow direction. This will enable accurate calculations for the modules but the entire heat exchanger including manifolds and distribution areas are not included. The idea of streamwise periodic flow and heat transfer was introduced by Patankar *et al.* (1977) and has been successfully followed up by many investigators. Nowadays, there are a number of software packages available and CFD codes are introduced to various extent in industries. To be successful in application of CFD methods for engineering tasks, it is required that experience and understanding exist for the physics of transport phenomena and fundamentals of numerical algorithms and turbulence modeling. Without these it is unlikely that the user will benefit extensively from the available commercial codes. Awarenesses of shortcomings and limitations are also important for successful application.

3.1 Governing equations

All the governing differential equations of mass conservation, transport of momentum, energy and mass fraction of species can be cast into a general partial differential equation as (Versteeg and Malalasekera, 2007; Ferziger and Peric, 1996):

$$\frac{\partial\, \rho\phi}{\partial\, t} + \frac{\partial}{\partial\, x_j}\rho\phi u_j = \frac{\partial}{\partial\, x_j}\left(\Gamma\frac{\partial\phi}{\partial\, x_j}\right) + S \qquad (13)$$

where ϕ is an arbitrary dependent variable, e.g. the velocity components, temperature, etc. Γ is the generalized diffusion coefficient, and S is the source term for ϕ. The general differential equation consists of four terms. From the left to the right in equation (13), they are called the unsteady term, the convection term, the diffusion term and the source term.

3.2 Turbulent flows

The most common turbulence models for industrial applications are classified as:

- zero-equation models;
- one-equation models;
- two-equation models;
- Reynolds stress models;
- algebraic stress models (ASM); and
- large eddy simulations (LES).

The three first models in this list account for the turbulent stresses and heat fluxes by introducing a turbulent viscosity (eddy viscosity) and a turbulent diffusivity (eddy diffusivity). Linear and nonlinear models exist (see, Pope, 2000; Wilcox, 2002; Durbin and Shih, 2005). The eddy viscosity is usually obtained from certain parameters representing the fluctuating motion. In two-equation models, these parameters are determined by solving two additional differential equations. However, one should remember that these equations are not exact but approximate and involves several adjustable constants. Models using the eddy viscosity and eddy diffusivity approach are isotropic in nature and cannot evaluate non-isotropic effects. Various modifications and alternate modelling concepts have been proposed. Examples of models of this category are the k-ε, and k-ω models in high or low Reynolds number versions as well as in linear and non-linear versions. A lately popular model is the so-called V2F model introduced by Durbin (1995). It extends the use of the k-ε model by incorporating near-wall turbulence anisotropy and non-local pressure-strain effects, while retaining a linear eddy viscosity assumption. Two additional transport equations are solved, namely one for the velocity fluctuation normal to walls and another one for a global relaxation factor.

In Reynolds stress equation models (RSM), differential equations for the turbulent stresses (Reynolds stresses) are solved and directional effects are naturally accounted for. Six modelled equations (i.e. not exact equations) for the turbulent stress transport are solved together with a model equation for the turbulent scalar dissipation rate ε. RSM models are quite complex and require large computing efforts and for this reason are not widely used for industrial flow and heat transfer applications.

ASM and explicit such (EASM) present an economic way to account for the anisotropy of the turbulent stresses without solving the Reynolds stress transport equations. One idea is that the convective and diffusive terms are modelled or even neglected and then the Reynolds stress equations reduce to a set of algebraic equations.

For calculation of the turbulent heat fluxes most commonly a simple eddy diffusivity concept is applied. The turbulent diffusivity for heat transport is then obtained by dividing the turbulent viscosity by a turbulent Prandtl number. Such a model cannot account for non-isotropic effects in the thermal field but still this model is frequently used in engineering applications. There are some models presented in the literature to account for non-isotropic heat transport, e.g. the generalized gradient diffusion hypothesis and the WET method. These higher order models require that the Reynolds stresses are calculated accurately by taking non-isotropic effects into account. If not, the performance may not be improved. In addition, partial differential equations can be formulated for the three turbulent heat fluxes but numerical solutions of these modelled equations are rarely found. Further details can be found in e.g. Launder (1988).

The LES is a model where the time-dependent flow equations are solved for the mean flow and the largest eddies while the effects of the smaller eddies are modelled. The LES model has been expected to emerge as the future model for industrial applications but it still limited to relatively low Reynolds number and simple geometries. Handling wall-bounded flows with focus on the near wall phenomena like heat and mass transfer and shear at high Reynolds number present a problem due to the near-wall resolution requirements. Complex wall topologies also present problem for LES.

Nowadays, approaches to combine LES- and RANS-based methods have been suggested.

3.2.1 Wall effects. There are two standard procedures to account for wall effects in numerical calculations of turbulent flow and heat transfer. One is to employ low Reynolds number modelling procedures, and the other is to apply the wall function method. The wall functions approach includes empirical formulas and functions linking the dependent variables at the near-wall cells to the corresponding parameters on the wall. The functions are composed of laws of the wall for the mean velocity and temperature, and fomulas for the near-wall turbulent quantities. The accuracy of the wall function approach is increasing with increasing Reynolds numbers. In general, the wall function approach is efficient and requires less CPU time and memory size but it becomes inaccurate at low Reynolds numbers. When low Reynolds number effects are important in the flow domain the wall function approach ceases to be valid. So-called low Reynolds number versions of the turbulence models are introduced and the molecular viscosity appears in the diffusion terms. In addition damping functions are introduced. Also so-called two-layer models have been suggested where the transport equation for the turbulent kinetic energy is solved while an algebraic equation is used for, e.g., the turbulent dissipation rate.

3.3 The finite volume method

In the finite volume method (FVM), the domain is subdivided into a number of so-called CVs. The integral form of the conservation equations are applied to each CV. At the centre of the CV a node point is placed. At this node the variables are located. The values of the variables at the faces of the CVs are determined by interpolation. The evaluation of the surface and volume integrals is carried out by quadrature formulas. Algebraic equations are obtained for each CV. In these equations values of the variables for neighbouring CVs appear.

The FVM is very suitable for complex geometries and the method is conservative as long as surface integrals are the same for CVs sharing boundary.

The FVM is a popular method particularly for convective flow and heat transfer. It is also applied in several commercial CFD-codes. Further details can be found in Versteeg and Malalasekera (2007) and Ferziger and Peric (1996).

The CV method is illustrated in Figure 4 and the governing equation is integrated over this volume element.

A formal integration of equation (13) (omitting the unsteady term) is:

$$\oiiint_V \frac{\partial \rho U_j \phi}{\partial x_j} dV \oiiint_V \frac{\partial}{\partial x_j}\left(\Gamma_\phi \frac{\partial \phi}{\partial x_j}\right) dV + \oiiint_V S_\phi dV. \tag{14}$$

Applying the divergence theorem gives:

$$\oiint_S \rho\phi \vec{U} \cdot d\vec{S} = \oiint_S \Gamma_\phi \nabla\phi \cdot d\vec{S} + \oiiint_V S_\phi dV. \tag{15}$$

3.3.1 Number of grid points and control volumes. The widths of the CVs do not need to be constant nor do the successive grid points have to be equally spaced. Often it is desirable to have a uniform grid spacing. Also, it is required that a fine grid is employed where steep gradients appear while a coarser grid spacing may suffice where slow variations occur. The various turbulence models require certain conditions on the grid structure close to solid walls. The so-called high and low Reynolds number versions of these models demand different conditions.

In general, it is recommended that the solution procedure is carried out on several grids with different fineness and varying degrees of non-uniformity. Then it might be possible to estimate the accuracy of the numerical solution procedure.

3.3.2 Complex geometries. CFD – methods based on Cartesian, cylindrical or spherical coordinate systems have limitations in complex or irregular geometries. Using Cartesian, cylindrical and/or spherical coordinates means that the boundary surfaces are treated in a stepwise manner. To overcome this problem methods based on body-fitted or curvilinear orthogonal, non-orthogonal grid systems are needed. Such grid systems may be unstructured, structured or block-structured or composite. Because the grid lines follow the boundaries, boundary conditions can more easily be implemented.

There are also some disadvantages with non-orthogonal grids. The transformed equations contain more terms and the grid non-orthogonality may cause unphysical solutions. Vectors and tensors maybe defined as Cartesian, covariant, contravariant

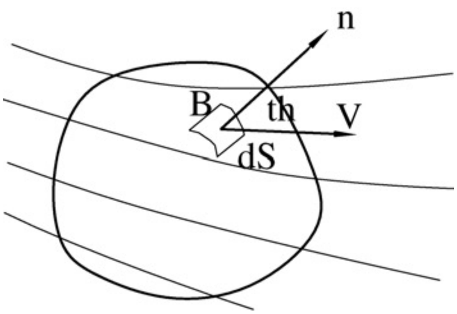


Figure 4.
Illustration CV method

and physical or non-physical coordinate-oriented. The arrangement of the variables on the grid affects the efficiency and accuracy of the solution algorithm.

Grid generation is an important issue and today most commercial CFD-packages have their own grid generators but also several grid generation packages, compatible with some CFD – codes, are available. The interaction with various CAD – packages is also an important issue today.

Various grid opportunities are shown in Figure 5.

3.3.3 Discretization. An arbitrary CV is shown in Figure 6. To solve equation (15), discretization summation over all surfaces is carried out. This is illustrated below.

In the discretized equation, the following terms need to be determined:

- convection flux C_f;
- diffusion flux D_f; and
- scalar value at a face Φ_f.

The convection flux C_f is commonly treated by the so-called Rhie and Chow interpolation scheme as conventional linear interpolation results in checker-board problem. One then has for velocity at the east face:

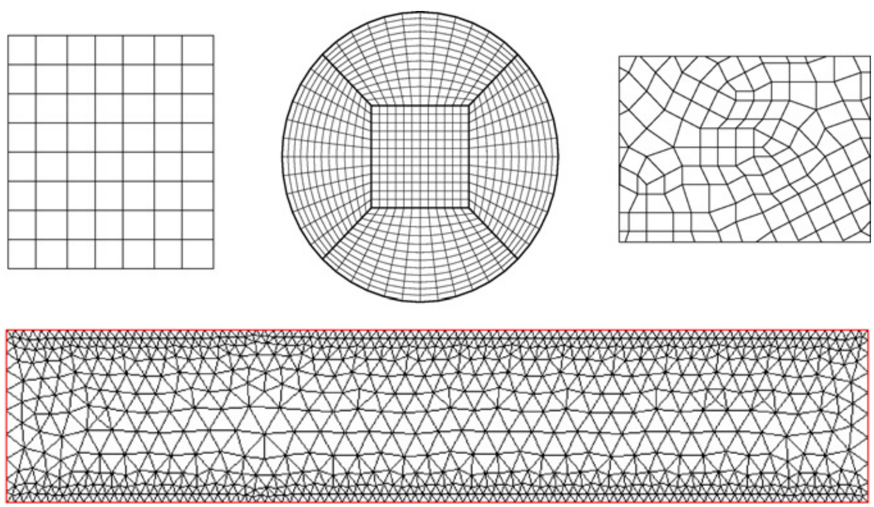
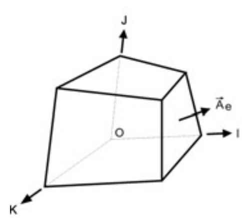


Figure 5.
Various grids

Notes: (a) Cartesian grid; (b) body fitted grid; (c) unstructured grid; and (d) unstructured grid with wall boundary layers



$$\oiint_S \phi\rho\vec{U}\bullet d\vec{S} = \oiint_S \Gamma_\phi \nabla\phi\bullet d\vec{S} + \iiint_V S_\phi dV$$

$$\sum_{f=1}^{nf} \phi_f C_f = \sum_{f=1}^{nf} D_f + S_\phi \Delta V$$

Figure 6.
Arbitrary CV and illustration of the integration across this volume element

$$U_e = f_x U_E + (1 - f_x) U_P + \frac{-(P_E - P_P)\Delta V}{|\overrightarrow{PE}|(a_P)_e}. \tag{16}$$

The diffusion flux D_f includes cross diffusion for non-orthogonal grids and the non-orthogonal part is put in the source term according to:

$$D_f = \Gamma_f \frac{\phi_1 - \phi_0}{ds} \frac{A \cdot A}{A \cdot \hat{e}_s} + \Gamma_f \left(\overline{\nabla\phi} \cdot A - \overline{\nabla\phi} \cdot \hat{e}_s \frac{A \cdot A}{A \cdot \hat{e}_s} \right). \tag{17}$$

The value of a scalar quantity at a face Φ_f can be calculated by different discretization schemes, e.g. Upwind, Hybrid, Central difference, QUICK, Power-Law, etc.

The pressure equation is constituted from the continuity equation and commonly a pressure correction is introduced. Then the velocity-pressure coupling can be handled by the algorithms SIMPLE, SIMPLEC, PISO etc. Alternatively, a Poisson equation for the pressure might be solved.

4. Results from some applications

4.1 Engineering approach

An existing finned heat exchanger (for an air-conditioning unit) according to Figure 1(b) was analysed. Hot water at 82 °C was used to heat an airflow by 15 °C. The heat exchanger area was 9.3 m^2 and the overall heat transfer coefficient U was estimated by formulas like equations (2) and (5) to be 230 W/m^{2K}. The outlet water temperature and the heat transfer rate were the output performance parameters. Then also the change in performance was considered as the airflow rate was varied. Because the heat exchanger geometry was given in this case, the ε-NTU method was preferable to apply.

4.2 Computational fluid dynamics approach

4.2.1 Ducts with bumps. As a first example a duct with bumps is considered. This type of duct appears in some rotary regenerative heat exchangers. The basic idea with introduction of bumps is to design corrugated ducts as indicated in Figure 7 for ducts with triangular cross section. The intention is that this corrugation should affect the flow field and introduce low Reynolds number turbulence and a swirling motion as sketched in Figure 7. At a certain distance downstream the corrugation element, the turbulence and the swirling motion will be attenuated and gradually the intensity of the fluctuations will be reduced. Therefore, at a position upstream that where the complex flow pattern (strong secondary cross-sectional flow and separated flow) has been significantly weakened or disappeared, a new corrugation element is introduced to re-establish the violent and swirling like motion. CFD-calculations have been performed and a non-orthogonal structured grid was employed. Periodic conditions were imposed in the main flow direction. About 40,000 CVs were used, 30 × 60 CVs in the cross-sectional plane. The existence of a secondary flow was revealed and a result

Figure 7.
Conjectured flow pattern in a duct with bumps

is shown in Figure 8. It is obvious that a swirling motion is created by the bumps and the triangular cross section. The Reynolds number corresponding to the flow in Figure 8 is about 2,000. In the simulations a low Reynolds number k-ε model was used. The secondary motion exists also for laminar cases as it is partly geometry driven. It is found that the heat transfer is enhanced compared to a smooth duct but the pressure drop increase is high.

Further details can be found in Sunden (2002).

4.2.2 Automotive radiators. Radiators in automotive applications are often of the flat tube-and-fin type heat exchanger. A sketch of the tube-fin arrangement was shown in Figure 1(c). Usually the fins have louvers as indicated in Figure 1(c).

Brass tubes and copper fins are used in competition with aluminium heat exchangers. The copper fins are brazed to the brass tubes. A disadvantage of brazing is that due to diffusion of alloying elements, an alloyed zone may be produced near the joint. This zone has quite different mechanical and thermal properties than the base materials, i.e. copper and brass. In some cases, intermetallic compounds (e.g. nickel-phosphor, copper-zinc-tin, copper-tin) may be formed while in other cases the brazing might be so bad that air gaps appear in the joints. It is important to know how the thermal performance is affected.

Figure 9 shows a brazing joint where the brazing material has diffused considerably into the copper fin. Also two air gaps are found in the joint. Figure 10 shows a joint where intermetallic compounds have been created and also a crack at the joint boundary is found. Figure 11 depicts an even more severe brazing failure with a large air gap, cracks and intermetallic phases.

In this case, a numerical investigation of the heat transfer process from the tube surface through the brazing joint and the fin coupled with forced convection on the cooling air side was conducted. Figure 12 shows the general model of the geometry while Figure 13 shows a typical grid distribution. The grid with the CVs constitutes a block-structured mesh with non-orthogonal cells. The grid being used in the final calculations was refined so that the influence of the CV sizes was negligible.

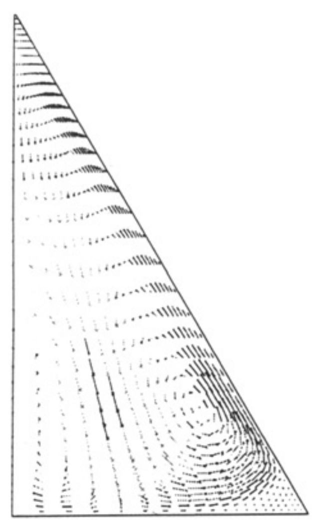


Figure 8.
Secondary flow velocity vectors in a cross-sectional plane midway over a corrugation element

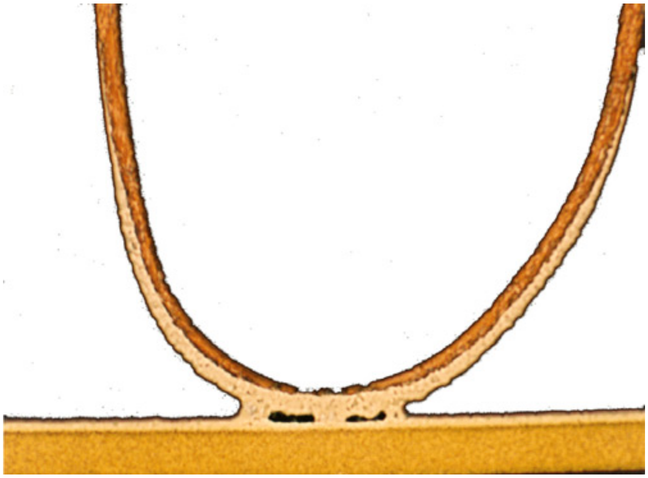


Figure 9. Brazing joint with air gaps

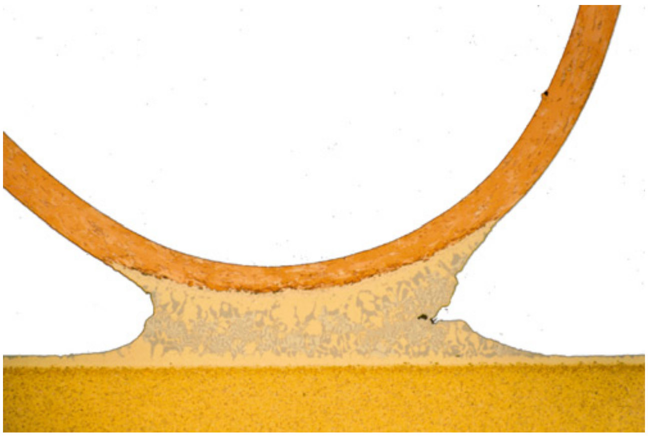


Figure 10. Brazing joint with intermetallic phases and a boundary crack

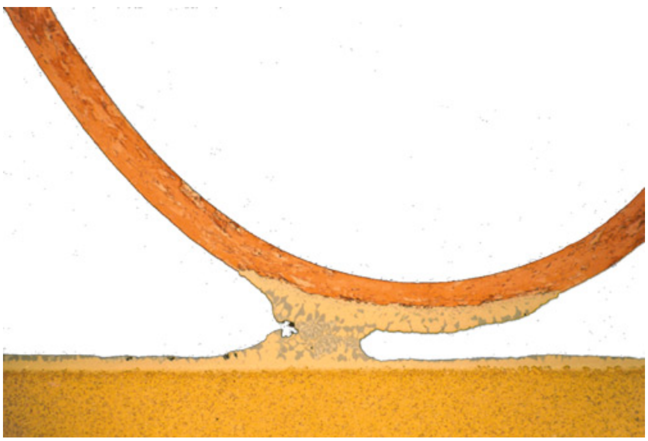


Figure 11. Poor brazing joint showing limited contact between fin and tube and a boundary crack

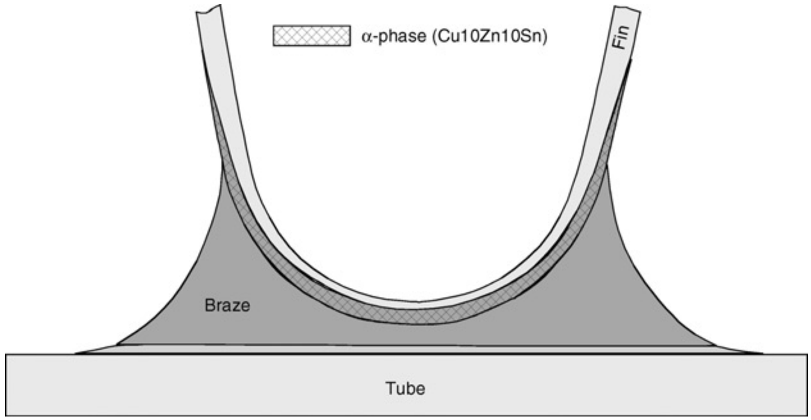


Figure 12.
Schematic model of an ideal brazing joint

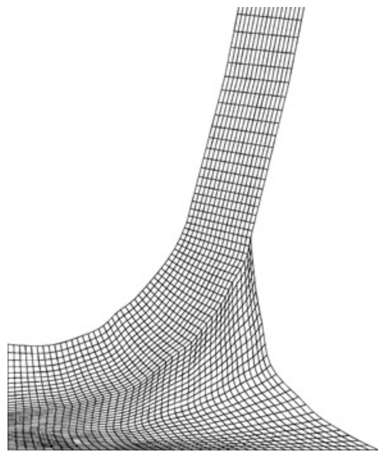
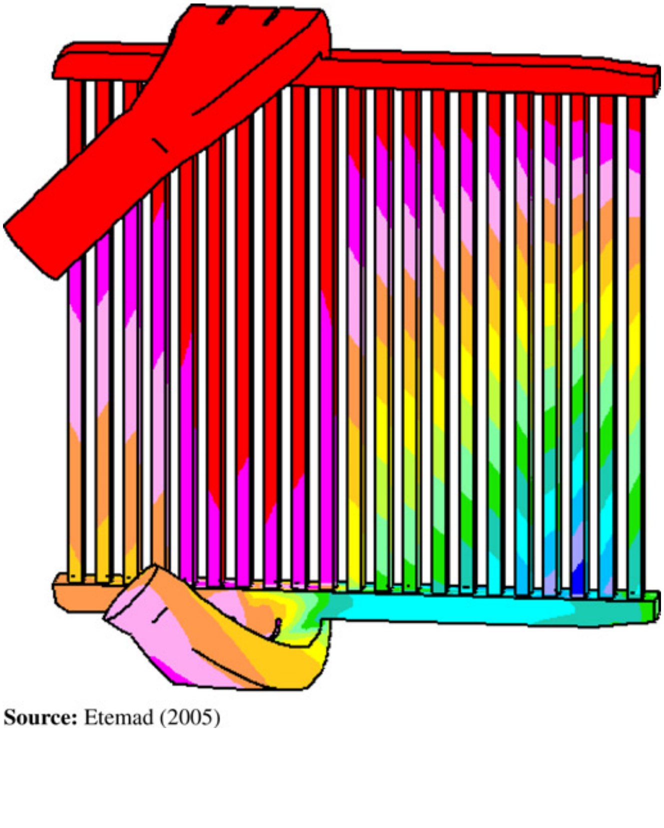


Figure 13.
Grid in a two-dimensional computational domain

Figure 14 shows a temperature distribution for a case with an air gap and it is found that a small temperature difference between the root and the tip exists but the local variation is strong. The two-dimensionality is strong. For this case the fin height was 5 mm, fin thickness 100 μm and the fin pitch 1 mm. The thermal conductivities were: copper fin 400 W/mK, braze material 45 W/mK and air gap 0.03 W/mK. The influence on the fin efficiency was not large but became more as the convective heat transfer coefficient was increased. Further details of this investigation are given in Sunden *et al.* (2000).

Another problem in radiators is flow maldistribution between the tubes in the core. To enable analysis of the manifolds and the complete core of tubes, a porous medium approach is appropriate. Figure 15 shows a typical result. The temperature at the inlet manifold (at the top) is uniform but due to the maldistribution of the flow (liquid), the temperature distribution is very uneven. It is evident that below the inlet, the flow rate is high and the temperature drop in the fluid is small. On the other hand, at the right and left edges where the flow rate is small a greater temperature change is found. By playing with various designs of the manifolds and connecting pipes, CFD can be used to evaluate the most proper design. Further details can be found in Etemad (2005).

Figure 14.
Temperature distribution in a copper fin and a brazing joint having an air gap



Source: Etemad (2005)

Figure 15.
Temperature distribution in a radiator due to flow maldistribution

Multilouvered fins are used quite frequently on the air side of radiators. Experimental investigations were carried out extensively in the past while during recent years CFD calculations have been increasingly applied. Besides predictions of overall properties like the friction factor and average Nusselt numbers, and studies of the influence of some geometry parameters, investigations of the flow structure have been presented. Usually assumptions of periodicity are adopted. Recent investigations have been presented in, e.g. Cui and Tafti (2002, 2003) and Sunden (2005a, b, 2007b, c).

5. Conclusions

A description of computational procedures for compact heat exchangers was provided.

Rating and sizing methods as well as CFD procedures for analysis of heat transfer and fluid flow for surfaces in heat exchanger applications were presented.

Examples to demonstrate the methods were provided.

Problems and difficulties were outlined.

It is found that computational heat transfer methods of various kind, complexity, etc. are useful tools if carefully handled. However, there are several constraints, difficulties and limitations to be aware of.

References

Cui, J. and Tafti, D.K. (2002), "Computations of flow and heat transfer in a three-dimensional multilouvered fin geometry", *International Journal of Heat and Mass Transfer*, Vol. 45, pp. 5007-23.

Cui, J. and Tafti, D.K. (2003), "Fin-tube junction effects on flow and heat transfer in flat tube multilouvered heat exchangers", *International Journal of Heat and Mass Transfer*, Vol. 46, pp. 2027-38.

Durbin, P.A. (1995), "Separated flow components with k-ε-v2 model", *AIAA Journal*, Vol. 33 No. 4, pp. 659-64.

Durbin, P.A. and Shih, T.I.-P. (2005), "An overview of turbulence modeling", in Sunden, B. and Faghri, M. (Eds), *Modeling and Simulation of Turbulent Heat Transfer*, WIT Press, Southampton, pp. 1-26.

Etemad, S. (2005), private communication.

Ferziger, J.H. and Peric, M. (1996), *Computational Methods for Fluid Dynamics*, Springer-Verlag, Berlin.

Incropera, F.P. and DeWitt, D.P. (1996), *Fundamentals of Heat and Mass Transfer*, 4th ed., J. Wiley & Sons, New York, NY.

Kaysand, W.M. and London, A.L. (1984), *Compact Heat Exchangers*, 3rd ed., McGraw-Hill, New York, NY.

Launder, B.E. (1988), "On the computation of convective heat transfer in complex turbulent flows", *ASME Journal of Heat Transfer*, Vol. 110, pp. 1112-28.

Patankar, S.V. and Spalding, D.B. (1974), "A calculation procedure for transient and steady state behavior of shell-and-tube heat exchangers", in Afgan, N.H. and Schlunder, E.U. (Eds), *Heat Exchanger Design Theory Source Book*, McGraw-Hill, New York, NY.

Patankar, S.V., Liu, C.H. and Sparrow, E.M. (1977), "Fully developed flow and heat transfer in ducts having streamwise-periodic variations of cross-sectional area", *ASME Journal of Heat Transfer*, Vol. 99, pp. 180-86.

Pope, S. (2000), *Turbulent Flows*, Cambridge University Press, New York, NY.

Shah, R.K. and Sekulic, D. (2003), *Fundamentals of Heat Exchanger Design*, J. Wiley & Sons, New York, NY.

Spalding, D.B. (2006), "Extending the boundaries of heat transfer", J.P. Hartnett Lecture, *13th International Heat Transfer Conference, Sydney*, available at: *CD ROM Proceedings, Begell House*.

Sunden, B. (2002), "On partially corrugated ducts in heat exchangers", ASME/IMECE 2002, available at: *CD-ROM Proceedings*, IMECE2002-39651.

Sunden, B. (2005a), "CFD in design and development of heat exchangers", ASME HT2005-72388.

Sunden, B. (2005b), "Computational heat transfer in heat exchanger analysis and design", ASME DETC2005-84746.

Sunden, B. (2007a), *Heat Transfer and Heat Exchangers, Kirk-Othmer Encyclopedia in Chemical Technology*, J. Wiley & Sons, New York, NY.

Sunden, B. (2007b), "Computational fluid dynamics in research and design of heat exchangers", *Heat Transfer Engineering*, Vol. 28 No. 11, pp. 898-910.

Sunden, B. (2007c), "Computational heat transfer in heat exchangers", *Heat Transfer Engineering*, Vol. 28 No. 11, pp. 895-97.

Sunden, B., Abdon, A. and Eriksson, D. (2000), "Influence of brazing on the heat transfer performance of fins in radiators", *ASME HTD-Vol. 365, PID-Vol. 4* (Advances in Enhanced Heat Transfer), pp. 35-40.

Versteeg, H.K. and Malalasekera, W. (2007), *An Introduction to Computational Fluid Dynamics, the Finite Volume Method*, 2nd ed., Pearson-Prentice-Hall, New York, NY.

White, F.M. (2003), *Fluid Mechanics*, 5th ed., McGraw-Hill, New York, NY.

Wilcox, D.C. (2002), *Turbulence Modeling for CFD*, 2nd ed., DCW Industries, La Cañada, CA.

Corresponding author
Bengt Sunden can be contacted at: bengt.sunden@energy.lth.se