American Journal of Engineering Science and Technology Research
Vol. 1, No. 9, December 2013, PP: 156 -166, ISSN: 2327-8269 (Online)
Available online at www.ajestr.com

Research article

Advances in Numerical Modeling of Heat Exchanger Related Fluid Flow and Heat Transfer

Arup Kumar Borah[1], Pramod Kumar Singh[2] and Prince Goswami [3]

[1]Department of Mathematics, R. G. Baruah College
Fatasil Ambari, Guwahati-781025, India
E-mail: borah.arup@yahoo.com
Telephone No: +9103612360084
[2] Department of Mathematics , University of Allahabad
Allahabad, Uttar Pradesh, Pin 211002, India
E-mail, pramod_ksingh@rediffmail.com
[3]Research Fellow, University Grants Commission
Bahadur Shah Zafar Marg, New Delhi – 110002, India

Abstract

The rapid development of numerical solution methods for non-linear differential equations and advances in computers during recent years have gradually enabled predictions of flow and temperature fields as well as associated heat fluxes and stresses in engineering applications. However, turbulence modeling still presents a problem as accurate reliable predictions of flow separation; reattaching, impinging and recirculating flow regions are required. In addition, experimental investigations are needed to verify the computations.

In several circumstances numerical investigations might be useful supplements to experimental testing methods, e.g., in providing details of local phenomena and fundamental mechanisms. For heat exchangers for both laminar and turbulent flows are of important and in addition the geometry is commonly more or less complex and sometimes of small dimensions. This paper considers current CFD methods including turbulence modeling, associated problems and limitations as well as providing examples of applications of CFD in a variety of heat exchanger problems. Narrow ducts of complex geometries will be of main concern. These will include wavy ducts, cross-corrugated surfaces, finned surfaces, and ducts with bumps.

Key Words: Heat exchanger, modeling, CFD, fluid flow, heat transfer

---

1. Introduction

Heat exchangers are device which commonly are used to transfer heat between two or more fluids. Many types of heat exchangers exist and have been developed and used in heat and power plants, refrigeration units, air-conditioning units, process industries, gas turbines systems, automotive applications, electronics cooling *etc*., [1].

In most heat exchangers, the heat transfer is due to convection and conduction from a hot to a cold fluid which are separated by solid walls. Design and sizing of heat exchangers involve many complex procedures. The total amount of heat transferred, pressure drops, performance efficiency, manufacturing and operating costs are important in the final design. In some cases the overall cost is important while in other applications weight and size are the most vital factors. Moreover, as a heat exchanger is designed the convective heat transfer coefficients between fluids and walls are important. Commonly the hot and cold fluids are flowing in channels of various designs. Thus internal channel flow is very important in the field of heat exchangers. These heat transfer coefficients are dependent on the flow velocity, fluid properties, channel cross section geometry and size, and duct length- coefficients are available [2, 3], for single phase flow at laminar, turbulent or transitional conditions. In developments of compact heat exchangers, it is important to decrease the heat transfer surface area. However, if still the same of higher amount of heat flow rate prevails, the only way out is to improve the heat transfer coefficients and thus reduces the thermal resistances on the hot and cold sides. Thus the concept of enhanced heat transfer [4] is very relevant for research and development of efficient heat exchangers. New surface and surface modifications like cross corrugated surface, dimples, ribbed surfaces, offset strip fins, louvered surfaces, inserts, wavy ducts, ducts with bumps *etc*. are being developed extensively.

There are several reasons why heat transfer and heat exchangers play a key role in the reduction of greenhouse gas emissions and for achieving a sustainable development [5, 6]. In new advanced techniques to reduce energy consumption and improving power conversion efficiencies, introduction of fuel cells, usage of renewable energy sources, application of exhaust gas recirculation and cooling etc. all call for heat transfer analysis and introduction of heat exchangers. Attempts to provide efficient, compact and cheap heat exchangers are indeed challenging. To achieve these both theoretical and experimental investigations must be conducted and advanced and advanced modern methods must be adopted. The present paper focus on modeling approaches and how such ones can be used in the research and development of heat exchangers. The paper briefly reviews current CFD methods for single phase flows including turbulent modeling, associated problems and limitations as well as providing examples of CFD applications in a variety of heat exchangers problems. Narrow ducts of complex geometries will be of main concern. Wavy ducts, cross-corrugated surfaces extended surfaces fins; ducts with bumps *etc.* will be included. Results from commercially available computer codes and in-house codes are presented.

2. Governing Equations

All the governing differential equations of mass conversation, momentum, energy and mass fraction of species can be cast into a general partial differential equation as

$$\frac{\partial\rho\emptyset}{\partial t}+\frac{\partial\rho\emptyset\, u_j}{\partial xj}=\frac{\partial}{\partial xj}\left(\Gamma\frac{\partial\emptyset}{\partial x_j}\right)+\ S \qquad (1)$$

Where ϕ represents the an arbitrary dependent variable - the velocity components, temperature *etc*., and Γ indicates the generalized diffusion coefficient, and S is the source term for ϕ. The general differential equation consists of four terms. From left to the right in Eq. (1), they are called unsteady term, the convection term, the diffusion term and the source term.

3. Modeling of Turbulent Flows

In heat transfer equipment like heat exchangers both laminar and turbulent flows are of interest. While laminar convective flow and heat transfer can be simulated by Eq. (1) (or transformed variants for arbitrary geometries), turbulent flow and heat transfer normally require modeling approaches in addition to Eq. (1) and this section gives a brief introduction to the modeling of turbulent flows.

The instantaneous mass conservation, momentum and energy equations form a closed set of five unknowns u, v, w, p and T. However, the computing requirements in terms of resolution in space and time for direct solution of the time dependent equations of fully turbulent flows at high Reynolds number (so called DNS calculations) are enormous and major developments in computer hardware and needed. Thus DNS is more viewed as a research tool for relatively simple flows at moderate Reynolds number. Meanwhile practicing thermal engineers need computational procedures supplying information about the turbulent processes, but avoiding the need to predict effects of every eddy in the flow.

This calls for information about the time-averaged properties of the flow and temperature fields (e.g., mean velocities, mean stresses, mean temperatures etc.). Commonly a time-averaging operation called Reynolds decomposition is carried out. Every variable is then written as a sum of time-averaged value and a superimposed fluctuating value. In the governing equations additional unknown appears, six for the momentum equations and three for the temperature field equation. The general variable is written as

$$\phi = \phi^{+} + \phi^{'} \qquad (2)$$

The additional terms in the differential equations have the forms

$$-\rho\, u'_i u'_j \text{ and } \rho\, c_p u'_i T \qquad (3)$$

and are called turbulent stresses and turbulent heat fluxes, respectively. Now, the task of turbulence modeling is to provide procedures to predict the additional unknowns, i.e., the turbulent stresses and turbulent heat fluxes with sufficient generality and accuracy. Methods based on the Reynolds averaged equations are commonly termed as RANS (Reynolds averaged Navier-Stokes equations) method.

4. Types of Models

The most common turbulence models for industrial applications are classified as

(a) zero-equation models
(b) one-equation models
(c) two-equation models
(d) Reynolds-equation models
(e) algebraic-equation models
(f) large-eddy simulation

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 non-linear models exists, the eddy viscosity is usually obtained from certain parameters representing the fluctuating motion. On the other hand, in two equation models, these parameters are determined by solving two additional differential equations. Models using the eddy viscosity and eddy diffusivity approach are isotropic

In nature and cannot evaluate non-isotropic effects. Various modifications and alternate modeling concepts have been proposed and examples 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.

In Reynolds stress equation models (RSM) differential equations for the turbulent stresses (Reynolds stresses) are solved and directional effects are naturally accounted for. Six modeled 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. Algebraic stress models (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 modeled or even neglected and then the Reynolds stress equations reduce to a set of algebraic equations.

The LES is model where the time-dependent flow equations are solved for the mean flow and largest eddies while the effects of the smaller eddies are modeled. 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 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.

4.1 Wall Effects

But 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 modeling procedures, and the other is to apply to wall function method. In general wall the wall function approach is efficient and requires less CPU time and memory size but it becomes inaccurate at low Reynolds numbers, [9].

5. Numerical Methods for Solution of Partial Differential Equations

In addition, some methods established for numerical solution of the governing equations of fluid flow and heat transfer problems. These are, finite volume method (FVM), the finite element method (FEM), the finite volume method (FVM), the finite difference method (FDM), the control volume finite element method (CVFEM) and the boundary element method (BEM)and the finite volume method will be considered more extensively. The Finite Difference method is the oldest method and is also easiest method to apply for problems with simple geometries. The computational domain is covered by a grid. Taylor series expansion or polynomial fitting is used to approximate the derivatives of the variables with respect to coordinates at each grid point [10].

5.1 Finite Volume Method

In this FVM to domain is subdivided into a number of so-called control volumes. The integral form of the conservation equations are applied to each control volume. At the center of the control volume a node point is placed. At this node the variables are located. The values of the variable at the faces of the control volumes are determined by interpolation. The evaluation of the surface and volume integrals are carried out by quadrature formulas. Algebraic equations are obtained for each control volume. In these equations values of the variables for neighbouring control volumes appear. The Finite volume method is a very suitable for complex geometries and the method is conservative as long as surface integrals are the same for control volumes sharing boundary. In addition the FVM is a popular method particularly for convective flow and heat transfer. It is also applied in several commercial CFD-codes [11-12]. The Finite Element Method has a good ability to handle complex geometries, Fletcher [13] and Reddy and Gartling [14].

5.2 Control Volume Finite Volume method and Boundary Element method

The control volume finite element method is a hybrid method between the FEM and FVM methods. In a two-dimensional case the domain is divided into triangular elements. The nodes are located at the vertices. Any variable is assumed to vary linearly within the element. The control volumes are formed around the nodes by joining the centroids of the elements and mid points on element edges. The conservation equations in integral form are applied to the control volumes as outlined for the FVM, Masson *et al.,* [15]. Moreover, the boundary element method (BEM) basically transforms the governing equations to boundary integrals which are to be solved numerically. For heat conduction problems it is well suited but becomes more complicated for convective flow and heat transfer, Power *et al.,* [16].

6. Commercial Computer Codes

Several industries and companies worldwide are nowadays using commercially available so-called CFD-codes (CFD-computational fluid dynamics) for simulation of flow and heat transfer, topics in heat exchangers, investigations on enhanced heat transfer, electronics cooling, gas turbine heat transfer *etc*. Among these codes – FLUENT, CFX, STAT-CD, FIDAP, ADINA, CFD2000, PHOENICS and others. However, to successfully apply such codes and to understand the fundamental concepts of computational methods.

7. Illustration of the Finite Volume method for a two dimensional case

American Journal of Engineering Science and Technology Research
Vol. 1, No. 9, December 2013, PP: 156 -166, ISSN: 2327-8269 (Online)
Available online at www.ajestr.com

Considering the general Eq.(1) in two dimensions. With a Cartesian coordinate system a rectangular grid is shown in Fig. (1). The grid points are denoted by upper case letters while the control volume faces appear with lower cases letters. Eq.(1) integrated over the control volume. By applying the mass conservation equation($\emptyset = 1,\ \Gamma = 0, S = 0$), the discretitized form of Eq.(1) becomes

$$a_P\, \phi_P = a_E\, \phi_E + a_w\, \phi_w + a_N\, \phi_N + a_S\, \phi_S + b \tag{4}$$

the coefficients a_P,………………,. a_N depend on the chosen difference scheme to handle the convection-diffusion terms. In general we may write

$$a_E = D_e A(|P_e|) + \max(-F_e, 0) \tag{5}$$

$$a_W = D_w A(|P_w|) + \max(F_w, 0) \tag{6}$$

$$a_N = D_n A(|P_n|) + \max(-F_n, 0) \tag{7}$$

$$a_S = D_s A(|P_s|) + \max(F_s, 0) \tag{8}$$

and

$$a_P = a_E + a_W + a_N + a_S + a_P^0 - S_P \Delta x\, \Delta y \tag{9}$$

$$a_P^0 = \rho_P^0 \Delta x\, \Delta y / \Delta\tau \tag{10}$$

$$b = S_C\, \Delta x\, \Delta y + a_P^0\, \emptyset_P^0 \tag{11}$$

The D:s, F:s and P:s in Eqs. (5) - (8) are

$D_e = \Gamma_e \frac{\Delta y}{\Delta x_e}$, $D_w = \Gamma_w \frac{\Delta y}{\Delta x_w}$, $D_n = \Gamma_n \frac{\Delta x}{\Delta y_n}$,

$D_S = \Gamma_S \frac{\Delta x}{\Delta y_S}$

$P_e = \frac{F_e}{D_e}$, $P_W = \frac{F_W}{D_W}$, $P_n = \frac{F_n}{D_n}$, $P_s = \frac{F_S}{D_S}$

$F_e = (\rho u)_e\, \Delta y,\ F_w = (\rho u)_w\, \Delta y,\ F_n = (\rho u)_n\, \Delta x, F_s = (\rho u)_s \Delta x$

The function $A(|P|)$ has different forms depending on the selected difference scheme. **Table 1** presents some examples:

Table 1: The function $A(|P|)$

$A(\lvert P\rvert)$	Difference scheme
1-0.5 ($\lvert P\rvert$)	CDS-central difference scheme
1	UDS-upstream scheme
Max (0, 1-0.5 ($\lvert P\rvert$))	Hybrid scheme
Max (0, 1-0.1$\lvert P\rvert^5$)	Power law scheme

If higher- order accurate scheme like QUICK, van Leer *etc*., are used more complicated formula then Eqs. (8)-(14) appear, i.e., additional grid points show up and the expressions for the coefficients change accordingly.

7.1 Source Term

The source term S may be depend on the variable ϕ and in the discretized equation it is desirable to account for such dependence. Commonly the source term is expressed as a linear function of ϕ. At the grid point P, S is then written as

$$S = S_c + S_p\, \phi \tag{12}$$

The linearization procedure above was used as the coefficients in Eq.(7) has been discussed.

7.2 Solution of the discretized equations

The discretized equations have the form of Eq. (7) with the - values at the grid point as unknown and for boundaries not having fixed - values, the boundaries values can be eliminated by using given or fixed conditions of the fluxes as such boundaries. Gauss elimination is a so called direct method to solve algebraic equations. For one-dimensional cases the coefficients form a tridiagonal matrix and an efficient algorithm called the Thomas algorithm or tridiagonal matrix algorithm (TDMA) is achieved. A popular method is a line-by-line technique combined with a block correction procedure. The equation along the chosen line are solved by the TDMA. Iterative methods are also needed because equations are non-linear and sometimes interlinked. In many situations, e.g., turbulent forced convection, the change in the value of from one iteration to another is so high that convergence in the iterative process is not achieved. To circumvent this and to reduce the magnitude of the changes, underrealxation factors (between 0 and 1) are introduced.

7.3 The pressure in the momentum equations and solution procedures of the momentum equations

In the momentum equations, a pressure gradient term appears in each coordinate direction (i.e., a source term *S*). If these gradients are known, the discretized equations for the flow velocities would follow the same procedure as for any scalar. However, in general the pressure gradients are not known but have to be found as part of the solution. Thus the pressure and velocity fields are coupled and the continuity equation (mass conservation equation) has to be used to develop a strategy. On the other hand, a special interpolation scheme has been used to calculate the velocities at the control volume faces. Most commonly the so-called Rhie-Chow interpolation methods is applied [17].

We have mentioned earlier that the velocity and pressure fields are coupled; thus a strategy has to be developed in the solution procedure of the momentum equations. The oldest algorithm is the SIMPLE (semi-implicit-method-pressure-linked-equations) algorithm. A pressure field p is guessed and then the momentum equations are solved for this pressure field resulting in a velocity field u_i^* . The pressure correction p' can be obtained. The velocity correction are related to the pressure corrections by

$$u_i = d_i \ \Delta p_i' \tag{13}$$

As the solution proceeds the velocities and pressure are corrected according to

$$p = p^* + p' \tag{14}$$

$$u_i = u_i^* + u_i' \tag{15}$$

The the momentum equations are solved again but now with the corrected pressure (16) as the guessed pressure. Inaddition, new velocities are obtained and new pressure and velocity corrections are calculated. The whole process is repeated until convergence is obtained. On the other hand, there are other similar algorithms available today. SIMPLEC (SIMPLE-consistent) and SIMPLEX (SIMPLE-extended) are common. They differ from SIMPLE mainly in the expression for di in Eq. (15), Anderson et al. [18].

Another algorithm is PISO (pressure implicit splitting operators) [19], has become popular and originally it is pressure-velocity coupling strategy for unsteady compressible flow. Compared to SIMPLE it involves one predictor step and two corrector steps. But, another algorithm is SIMPLER (SIMPLE-revised). Here the continuity equation is used to derive a discretized equation for the pressure. The pressure correction is then only used to update the velocities through the velocity corrections.

7.4 Convergence

The solution procedure is in general iterative and then some criterion must be used to decide when a converged solution has been reached. One method is to calculate residuals R as

$$R = \sum_{NB} a_{NB} \quad \phi_{NB} \ + \ b - a_P \, \phi_P \tag{16}$$

for all variables; NB indicates neighbouring grid points *e.g.*, E, W, N, S. If the solution is converged, R = 0 everywhere. Practically, it is often stated that the largest value of the residuals [R] should be less than a certain number. If this is achieved the solution is said to be converged.

7.5 Number of grid points and control volumes

The widths of the control volumes do not need to be constant nor do the successive grid points have to be equally spaced. Furthermore, it is desirable to have uniform grid spacing. Also it is required that a fine grid is employed where steep gradients appear while coarser grid spacing may suffice where slow variations occur. The various turbulence models require certain conditions on the grid structures close to solid walls. The so-called high and low Reynolds number versions of these models demand different conditions. On the other hand, it is recommended that the solution procedure is carried out on several grids with different fineness and varying degrees of non-uniformity. It might be possible to estimate the accuracy of the numerical solution procedure.

8. Complex Geometries

CFD-methods based on Cartesian or cylindrical coordinate systems have limitations in complex or irregular geometries. Applying Cartesian and/ or cylindrical coordinates means that the boundary surfaces are treated in a stepwise manner. To overcome these problems methods based on body –fitted or nonorthogonal grid systems are needed. Such grid systems may be unstructured, structured or block-structured. Because the grid lines follow the boundaries, boundary conditions can more easily be implemented. On the other hand, there are also some disadvanges with non-orthogal grids. The transformed equations contain more terms and the grid-nonorthogonality may cause unphysical solutions. In other words, the arrangements of the varaiables on the gtid affect 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 (computer-added design) –packages also an important issue today treating complex geometries [20, 21].

9. Results and Discussion

In this study some results from CFD analysis of heat transfer and fluid flow in various heat exchanger applications have been studied.

10. Cross-corrugates surfaces

Plate-and-frame heat exchangers (PHE) are in used in many different processes in a broad range of temperatures with a variety of substances. In addition various types of PHEs exist. They are all assembled of plates with an embossed surface area enhancement pattern. Today, the most common pattern in use is the chevron or herringbone design. The embossed pattern results in a larger heat transfer surface than a flat plate, improves the plate stiffness and assures the channel gap. The geometry of corrugation is very important for thermal-hydraulic performance of the heat exchanger. Each plate has four corner ports which, in pairs, establish access to the narrow flow passages on their side of the plate. Usually there are also distribution areas between the ports and embossed plate area. Fig. (2) shows a plate with a chevron pattern. Plates are usually staked together in a symmetric or mixed arrangement as indicated in Fig. (3). The heat transfer enhancement in PHE:s is related to the plate characteristics and may be attributed to increased effective heat transfer area, small hydraulic increase effective heat transfer area, small hydraulic diameter flow channels and vortex or swirl flow generation. The inclination angle, corrugation amplitude, corrugation wavelength and the profile of the waviness are important for the local and overall heat transfer coefficients as well as pressure loses. The geometry investigated is consistent with typical commercially available plate heat exchangers. Fig. (4) shows a segment of the geometry of the corrugated plates. In the numerical solution approach, the smallest volume, for which periodic conditions can be realized, is attempted. The whole plate areas will consists of many such cells. Fig.(5) the most important dimensions are indicated, the pitch p, height b, angle 2β and plate thickness *t*. Again, Fig. (6) shows a side view of the unit cell, which can be considered as the plane of geometrical symmetry.

The specific geometric considered has the dimensions: b = 3.05.10-3 m, p = 1.07.10-2 and R = 2.3.10-3 m. For this geometry 32□16□ 32 control volumes were used in each block. But, in Reynolds number range was 900-20000. For this geometry experimental results were available for the average Nusselt number and the friction factor. These results, investigated by Alfa Laval AB, Lund were deduced from overall performance tests. The chosen number of the grids points was found appropriate as a balance between computational times and sufficient numerical accuracy was attempted for. The RNG k-ε model was applied. Water with Prandtl number 4.33 was the fluid. Fig.(7) shows the flow pattern, at mid plane parallel with the lower furrow axis, for the geometry. From the above observations it is noticed that the experimental values for the unit cell were achieved from overall data from which the inlet and outlet regions and the flow distribution areas were subtracted. Moreover, a certain additional degree of uncertainty is thus present in the experimental values presented in Fig. (8). Nevertheless, the computed results are promising and encouraging, [22, 23].

Acknowledgement

The authors would like to acknowledge the financial support from the University Grants Commission, New Delhi-110002, India, under F. No. 40-236/2011 (SR). The author is also greateful of Kambiz Vafai, University of California, Department of mechanical Engineering, Riverside CA 92521, Bourns Hall A363 for providing some relevant informations during the preparation of the manuscript.

Nomenclature

a	coefficient	[kg/ms]
A	function	[-]
b	coefficient	
c_p	specific heat	[J/kgK]
D	diffusion coefficient	[kg/ms]
F	mass flow rate	[kg/ms]
k	turbulent kinetic energy	[m^2/s]
P	Peclet number	….
p	pressure	[Pa]
p'	pressure fluctuation	[Pa]
R	Residual	….
S	Source term	….
S_C	constant part of S	…..
S_P	linear coefficient of S	…..
T'	temperature fluctuation	[K]
t	time	[sec]
u	velocity in x-direction	[m]
v	velocity in y-direction	[m]
u_i	velocity vector	[m/s]
x_j	Cartesian coordinates	[m]
Greek symbols		
Δx	step-size x direction	[m]
Δy	step-size y direction	[m]

$\Delta\tau$ step-size in time [s]
ε dissipation rate [m^2/s^3]
$\emptyset$ arbitrary variable …….
ϕ' fluctuating value of $\emptyset$ ……..
Γ diffusion coefficient [m^2/s]
ρ density [kg/m^3]

Subscripts

E, W, N, S east, west, north, south grid point respectively
e, w, n, s east, west, north, south face respectively

References

[1] R. K. Shah and D. P. Sekulic, Fundamental of Heat Exchangers Design, J. Wiley and Sons, Inc. New Jersey, 2003.

[2] R. K. Shah and A. L. London, Laminar flow Forced Convection in Ducts, Advances in Heat Transfer Supplement I, Academic Press, New York, 1978.

[3] S. Kakac, R. K. Shah and W. Aung, Handbook of Single – Phase Convective Heat Transfer, Wiley Interscience Publication, USA, 1987.

[4] R. L. Webb, Principles of Enhanced Heat Transfer, J Wiley & Sons, New York USA, 1994.

[5] B. Suden and L. Wang, Relevance of Heat Transfer and Heat Exchangers for GreenHouse Emmissions, in Advances in Heat Transfer Engineering , B. Suden, and J. Vilemas, Eds., pp. 101-112, Begell House, Inc. New York 2003.

[6] B. Suden and L. Wang, Relevance of Heat Transfer and Heat Exchangers for Development of Sustainable Energy Systems, AJTEC 2003, TED-AJ03-603, 2003. (CD-ROM proceeding).

[7] D. C., Wilcox, Turbulence Modeling for CFD, DCW Industries, INC., La Canada, California, 1993.

[8] P.A. Durbin and T.I-P, Shih, An overview of Turbulence Modeling, in Modeling and Simulation of Turbulent Heat Transfer, B. Suden and F. Faghri, Eds., pp. 1-26 WIT Press,, Southampton, UK, 2003. (to appear).

[9] R. Amano, Turbulent Heat Transfer in Corrugated Wall Channel, in Computer Simulations in Compact Heat Exchangers, Eds. B. Sudenand M. Faghri, Chaper 5 pp. 115-149, Computational MechanicsPublication Southampton, UK, 1998.

[10] G.D Smith, Numerical Solution of Partial Differential Equations, Oxford University Press, UK, 1978.

[11] S. V. Patankar, Numerical Heat transfer and Fluid Flow, Mc-Graw Hill Book Company, USA, 1980.

[12] H. K, Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics; the Finite Volume method, Longman Sci. & Techs. UK 1995.

[13] C.A.J Fletcher, Computational Techniques for Fluid Dynamics, Vol.I, Springer, Berlin, Germany 1991.

[14] J. N. Reddy and D. K. Gartling, The Finite Element Method in Heat Transfer and Fluid Dynamics, CRC Press, Boca Raton, Florida USA, 1994.

[15] C. Masso, H.J. Saabas and K.B. Baliga, Co located equal- order control-volume finite element method for two-dimensional axisymmetric incompressible flow, Int .J. Numer. Meth. Fluids, 18. pp. 1-26, (1994).

[16] H. Power and L. Wrobel, Boundary Integral Methods in Fluid Mechanics, Computational Mechanics Publications, Southampton, UK 1995.

[17] C. M. Rhie and W. L. Chow, Numerical Study of the turbulent flow past an airfoil with trailing edge separation, AIAA. J. 21, pp. 1525-1532, 1983

[18] D. A. Anderson, J. C. Tannehill and R.H. Pletcher, Computational Fluid Mechanics and Heat Transfer, Taylor & Francis, USA, 1997.

[19] R. I. Issa, Solution of the implicitly duiscretized fluid flow equations by operator-splitting, J. Comp. Phys. 62 pp. 45-65, 1986.

[20] B. Farhanieh, L. Davidson and B. Suden, Employment of second moment closure for calculation of recirculating flows in complex geometries with collocated variable arrangement, Int. J. Num. Methd. Fluids, 16, pp. 525-544, 1993.

[21] J. H. Ferzier and M. Peric, Computational methods for Fluid Dynamics, Springer Verlag, Berlin, Germany, 1996.

[22] B. Suden and I. Di Piazza, Numerical Analysis of Fluid flow and Heat Transfer in Plate-and- Frame Heat Exchangers, ASME HTD-Vol. 361--/PID-Vol. 3, pp. 287-293, 1998.

[23] B. Suden, Flow and Heat Transfer Mechanisms in Plate-and-Frame Heat Exchangers , in Heat Transfer Enhancements of Heat Exchangers , Eds. S. Kakac et al. pp. 185-206, Kluwer Academic Publishers 1999.