ELSEVIER

International Journal of Heat and Fluid Flow 21 (2000) 463–467

International Journal of HEAT AND FLUID FLOW

www.elsevier.com/locate/ijhff

A note on the solution of conjugate heat transfer problems using SIMPLE-like algorithms

Xi Chen *, Peng Han

Engineering Thermophysics Division, Department of Engineering Mechanics, Tsinghua University, Beijing 100084, People's Republic of China

Received 24 March 1999; accepted 3 March 2000

Abstract

Conjugate conduction and convection heat transfer problems are often solved numerically in a unitary computational domain containing both the solid and fluid regions. When SIMPLE-like algorithms are employed in the computation, one has to pay close attention to the peculiarity of the energy equation or to the problem how to ensure the continuity of heat flux at the fluid–solid interface. For this reason, two approaches, i.e., the 'pseudo-density' method and the 'pseudo-solid-specific-heat' method, are suggested, respectively, for the solution of the energy equation, and typical computed results are presented.

Keywords: Conjugate heat transfer; Numerical solution; SIMPLE-like algorithms

1. Introduction

Conjugate heat transfer problems are encountered in many practical applications, where heat conduction in a solid region is closely coupled with convection heat transfer in an adjacent fluid region. Typical examples include heat transfer enhancement with a finned surface, heat transfer in a cavity with thermally conducting walls or internal baffle, cooling of microelectronic chips, heat transfer in the workpiece for arc welding, and so on. Many papers have been published in the recent two decades concerning the numerical modeling of the conjugate heat transfer problems (e.g., Kim and Viskanta, 1985; Davalath and Bayazitoglou, 1987; Gonzalez et al., 1993; Choi and Kim, 1996; Ju and Chen, 1996).

A straightforward and time-saving approach for the solution of conjugate heat transfer problems is to employ a unitary computational domain containing both the solid and fluid regions. Nevertheless, it should be pointed out that if the SIMPLE-like algorithms (SIMPLE, SIMPLER, SIMPLEC, etc.) are employed in the computation for this case, one needs to pay close attention to the solution of energy equation in order to ensure the physical realism of the computed results. Any negligence in this aspect will result in erroneous results. This note aims at discussing this special problem.

In the conventional SIMPLE-like algorithms (Patankar, 1980), the following general governing equation is handled:

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho v_i\phi)}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\Gamma\frac{\partial\phi}{\partial x_i}\right)+S. \tag{1}$$

In addition, as one of the basic rules of discretization, the flux $-\Gamma(\partial\phi/\partial x_i)$ across the face separating any two adjacent control volumes must be represented by the same expression in the discretized equations for the two control volumes in order to ensure the physical realism and the overall balance of computed results. When unequal materials transport properties (thermal conductivity, diffusion coefficient, etc.) are involved in the computation, a harmonic mean of the 'diffusion coefficient' (Γ) was recommended to be adopted at the interface separating the two adjacent control volumes along with the evaluation of the derivative $(\partial\phi/\partial x_i)$ using a piecewise-linear profile (Patankar, 1980).

For the conjugate heat transfer problem, when the energy equation is solved together with other equations (e.g., continuity and momentum equations) using the standard SIMPLE-like algorithms, one may encounter some special problems concerning the peculiarity of the energy equation or concerning how to ensure the heat flux continuity at the solid–fluid interface. In order to clarify this point, in the following let us study such a simple conjugate heat transfer problem: the steady two-dimensional (2D) convection–conduction heat transfer across a planar interface separating the solid and fluid regions. To stress the problem point itself, it is assumed that there is no surface heat source, phase change or fluid dissipation (or suction) at the solid–fluid interface in the discussion. Solid or fluid properties are assumed to be constant, although the properties of the solid material may be quite different from those of the liquid.

* Corresponding author.
E-mail address: cx-dem@mail.tsinghua.edu.cn (X. Chen).

0142-727X/00/$ - see front matter
PII: S0142-727X(00)00028-X

Notation

C_p	specific heat
h	specific enthalpy
k	thermal conductivity
L	length
q	specific heat flux
S	source term
T	temperature
t	time
v_i, v_x, v_y	velocity components
x, y	Cartesian coordinate axes

Greeks

α	thermal diffusivity $[\alpha = k/(\rho C_p)]$
Γ	generalized diffusion coefficient
δ	width or thickness
θ	dimensionless temperature
μ	fluid viscosity
ρ	density
ϕ	dependent variable
Ψ	nominal flux $[\Psi = -\Gamma(\partial\phi/\partial x)]$

Subscripts

0	reference state
1, 2, 3, 4	control volume number, or medium index
f	fluid
s	solid; interface

2. Analysis

For the steady 2D conjugate heat transfer problem under study, the energy equation in a Cartesian coordinate system with temperature T as the dependent variable is

$$\frac{\partial(\rho v_x C_p T)}{\partial x} + \frac{\partial(\rho v_y C_p T)}{\partial y} = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + S_T \qquad (2)$$

for the fluid region. Noting that convection is absent in a solid, Eq. (2) with $v_x = v_y = 0$ is also applicable to the solid region. Fig. 1 plots a few typical control volumes separated by the solid–fluid interface, i.e., control volumes 1 and 3 in the solid side and control volumes 2 and 4 in the fluid side. When a uniform grid spacing is used in the computation and Eq. (2) is discretized for control volumes 1 and 2, the 'diffusion flux' or the heat flux $q_s = -[k(\partial T/\partial x)]_s$ across the fluid–solid interface separating the control volume 1 and the control volume 2 will be expressed as

$$q_s = -\frac{2k_1k_2}{(k_1+k_2)}\frac{T_2 - T_1}{x_2 - x_1}. \qquad (3)$$

Here the harmonic mean of the thermal conductivities (k_1 and k_2) and the piecewise-linear temperature profile have been employed to evaluate the heat flux at the interface in order to ensure the overall energy balance of the computed results (Patankar, 1980). Heat flux continuity at the interface can always be satisfied for this situation. However, it should be noted that Eq. (2) is somewhat different from the general form (1) of the governing equations. Namely, ρC_p on the left-hand side of Eq. (2) corresponds to ρ on the left-hand side of the general equation (1) if the dependent variable ϕ is taken to be temperature T. Ignoring this difference in programming may result in a serious error in the computational results if the standard SIMPLE-like algorithms are employed in the computation and all the governing equations are treated as the general form shown in Eq. (1). A simple but very efficient approach (Approach 1) employed to avoid this error is as follows: whenever the energy equation (2) is to be solved in each round of the iterations, the density ρ in Eq. (1) is temporarily substituted by the pseudo-density ρ^*, where $\rho^* = \rho C_p$ as seen by comparing Eq. (1) with Eq. (2). After the energy equation has been solved, the actual density ρ is re-used for the solution of other equations (momentum equations and pressure correction equation, etc.). For this situation, the density ρ, the specific heat C_p and the thermal conductivity k in Eq. (2) are their actual values in both the solid and fluid regions (solid density and specific heat do not affect the computed results due to the absence of the convection terms in the solid region). Since the pseudo-density ρ^* has been used in the solution of the energy equation (2), we can also call Approach 1 the pseudo-density method.

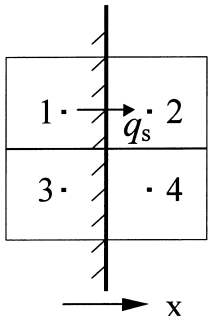

Fig. 1. Adjacent control volumes across the solid–fluid interface.

Many researchers preferred to employ the following form of the energy equation:

$$\frac{\partial(\rho v_x T)}{\partial x} + \frac{\partial(\rho v_y T)}{\partial y} = \frac{\partial}{\partial x}\left(\frac{k}{C_p}\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{k}{C_p}\frac{\partial T}{\partial y}\right) + \frac{S_T}{C_p}. \qquad (4)$$

The merit of Eq. (4) compared to Eq. (2) is that it is consistent with the general form (1) of the governing equations and thus Approach 1 is no longer needed. Eq. (4) is also applicable to both the fluid and solid regions. However, when the conjugate heat transfer problem is solved numerically using Eq. (4) in the unitary computational domain, one has to pay close attention to ensure the continuity of heat flux at the solid–fluid interface. The reason is as follows: when a uniform grid spacing is used in the computation and Eq. (4) is discretized for control volumes 1 and 2 shown in Fig. 1, the diffusion flux $\Psi_s = -[\Gamma(\partial T/\partial x)]_s$ across the face separating the control volume 1 and the control volume 2 will be expressed as

$$\Psi_s = -\left(\frac{k}{C_p}\frac{\partial T}{\partial x}\right)_s = -\frac{2k_1k_2}{k_1C_{p2} + k_2C_{p1}}\frac{T_2 - T_1}{x_2 - x_1}. \qquad (5)$$

Here the harmonic mean of k_1/C_{p1} and k_2/C_{p2} and the piecewise-linear temperature profile have also been adopted to evaluate the diffusion flux at the interface in order to ensure the overall balance of the computed results (Patankar, 1980). However, it should be noted that Ψ_s is only a nominal flux, its continuity at the interface cannot ensure the physical reality. Eq. (5) shows that both solid and fluid specific heats (C_{p1} and C_{p2}) affect the diffusion flux Ψ_s at the interface and thus affect the computed temperature field. However, because only steady heat conduction occurs in the solid region, the actual

temperature field should be independent of the solid specific heat. Hence, the computed temperature field based on (5) will be physically incorrect for the case $C_{p1} \neq C_{p2}$, although this fact was often ignored by some authors (e.g., Gonzalez et al., 1993; Ju and Chen, 1996). If great difference in the values of the solid and fluid specific heats is involved in the computation, appreciable discontinuity of the heat flux will appear at the solid–fluid interface and serious error will be introduced in the computational results, as we can see later on from the typical calculated results. A simple but very efficient approach (Approach 2) to avoid this error is as follows: the thermal conductivity in Eq. (4) is taken to be its actual value in the solid or fluid region, but the specific heat of the solid region is artificially set to be equal to that of the fluid region. As a result, the actual solid specific heat will not affect the computed temperature field, as desired, and Eq. (5) reduces to $\Psi_s = q_s/C_{pf}$, i.e., the heat flux continuity can be ensured at the fluid–solid interface. The reason for choosing the specific heat of fluid (C_{p2} or C_{pf}) instead of that of solid (C_{p1} or C_{ps}) in this approach is that the specific heat affects the results only through the convection term of the energy equation, and thus only the fluid specific heat is important. Since the specific heat in the solid region used here is not its actual value, Approach 2 can also be called as the 'pseudo-solid-specific-heat' method. Approach 2 is also applicable if the following energy equations are employed:

$$\frac{\partial(\rho v_x h)}{\partial x} + \frac{\partial(\rho v_y h)}{\partial y} = \frac{\partial}{\partial x}\left(\frac{k}{C_p}\frac{\partial h}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{k}{C_p}\frac{\partial h}{\partial y}\right) + S_h, \tag{6}$$

$$v_x\frac{\partial \theta}{\partial x} + v_y\frac{\partial \theta}{\partial y} = \alpha\left(\frac{\partial^2\theta}{\partial x^2} + \frac{\partial^2\theta}{\partial y^2}\right) + S_\theta. \tag{7}$$

When Eq. (6) is adopted, Approach 2 requires that C_p in Eq. (6) should be taken as the fluid specific heat everywhere and the specific enthalpy h should be related with the temperature by $h = h_0 + C_{pf}(T - T_0)$ for both the solid and fluid regions (h_0 is the specific enthalpy at the reference temperature T_0). On the other hand, when Eq. (7) is used, Approach 2 requires that the product of the density and specific heat of the fluid, i.e., $(\rho C_p)_f$, should be used in the evaluation of the thermal diffusivities for both the solid and fluid regions, i.e., $\alpha_s = k_s/(\rho C_p)_f$ and $\alpha_f = k_f/(\rho C_p)_f$.

3. Typical calculated results and discussion

In order to check the validity of the suggested approaches, let us examine a few typical examples. The first example is the conjugate heat transfer problem in a 2D counter-flow heat exchanger shown in Fig. 2. In a Cartesian coordinate system, this heat exchanger consists of two parallel flow passages with widths δ_1 and δ_3 separated by a metallic plate with thickness of δ_2. The outer walls of the flow passages are assumed to be adiabatic. The same properties and uniform inlet velocity and temperature profiles are assumed for the hot and cold fluids.

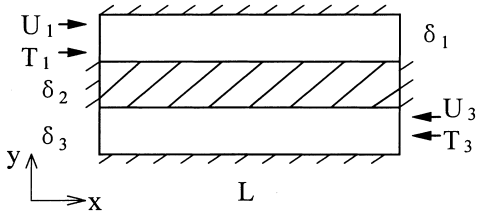

Fig. 2. A 2D counter-flow heat exchanger.

Parameters adopted in the computation are as follows: geometrical sizes $\delta_1 = \delta_2 = \delta_3 = 0.1$ m and $L = 1$ m; fluid parameters $U_1 = 0.2$ m/s, $T_1 = 800$ K, $U_3 = 0.1$ m/s, $T_3 = 300$ K, $\rho_f = 1000$ kg/m^3, $k_f = 10$ W/(m K), $C_{pf} = 25$ J/(kg K) and $\mu_f = 0.15$ kg/(m s); metallic plate properties $\rho_s = 8000$ kg/m^3, $k_s = 50$ W/(m K) and $C_{ps} = 500$ J/(kg K). In order to emphasize the effect of the solid/fluid specific-heat ratio, a much smaller value of the fluid specific heat has been employed here. The energy equation (4) or (2) and the continuity and momentum equations (as usual) are solved simultaneously using the SIMPLE algorithm, and grid points are taken to be 62×32 with uniformly distributed control volumes. Three different methods designated as Method A, Method B and Method C are used, respectively, in the computation. Here Method A means that k/C_p in Eq. (4) is the actual values for both the solid and fluid regions, i.e., $k/C_p = k_s/C_{ps}$ for the metallic plate, while $k/C_p = k_f/C_{pf}$ for the hot or cold fluid. Method B denotes that Approach 2 (or the pseudo-solid-specific-heat method) is used, i.e., k/C_P in Eq. (4) is $k/C_p = k_s/C_{pf}$ for the metallic plate, while $k/C_p = k_f/C_{pf}$ for the hot and cold fluids. Method C expresses that the energy equation (2) and Approach 1 (or the pseudo-density method) are employed. The computed temperature distributions within the 2D heat exchanger obtained using Method A and Method B are shown in Fig. 3 and Fig. 4, respectively. Method A predicts that unexpected large temperature drop occurs across the metallic partition plate, while the temperature variation within the hot or cold fluid is much smaller (Fig. 3). On the other hand, Method B predicts that more significant temperature difference occurs in the hot or cold fluid along the flow direction, while the temperature drop across the metallic plate is comparatively small (Fig. 4). The predicted results using Method C are almost identical to those shown in Fig. 4 obtained by the Method B and thus not plotted as a new figure. As expected,

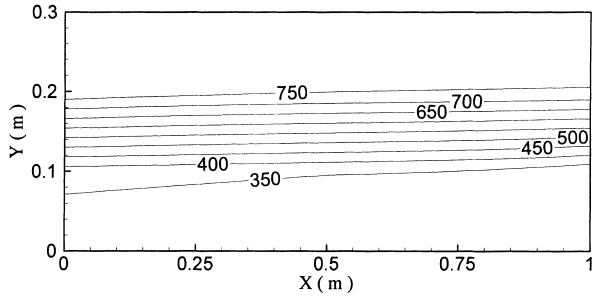

Fig. 3. The temperature distribution within the 2D counter-flow heat exchanger obtained using Method A (temperature in K).

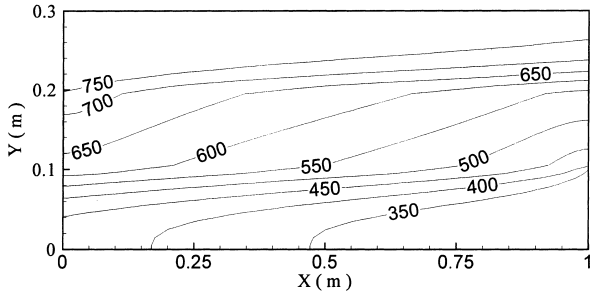

Fig. 4. The temperature distribution within the 2D counter-flow heat exchanger obtained using Method B (or Approach 2, temperature in K) or Method C.

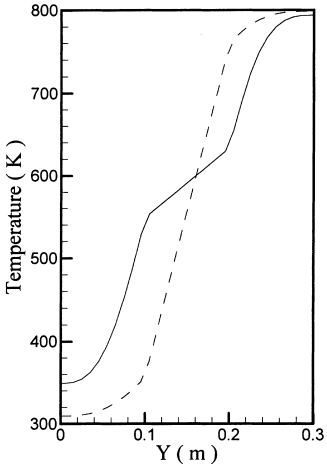

Fig. 5. The temperature profiles at $X = L/2$ for the 2D counter-flow heat exchanger. Dash line: Method A; solid line: Method B or Method C.

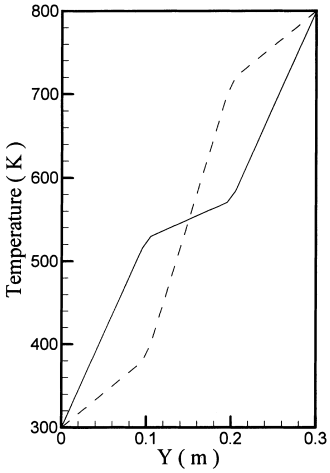

Fig. 6. The temperature profiles for pure heat conduction. Dash line: Method A; solid line: exact solution, Method B (or Approach 2) or Method C (or Approach 1).

quite different temperature profiles at the cross-section with $X = L/2$ are obtained by Method A (dash line) and Method B or Method C (solid line), as shown in Fig. 5. Since the value of the thermal conductivity of the metallic plate is five times as great as that of the fluid, it is obvious that the temperature drop within the metallic plate should be much less than that within the hot or cold fluid. Hence, the predicted results by Method A are physically incorrect (heat flux is obviously discontinuous at the fluid–plate interfaces), while Method B and Method C can predict reasonable temperature fields. The reason is that if Method A is used in the computation, $k_s C_{pf}/C_{ps}$, instead of the actual value k_s, is actually used as the effective thermal conductivity of the metallic plate, as shown by comparing Eq. (2) with Eq. (4), and thus the computed results are incorrect.

In order to check quantitatively the suggested approaches, let us examine another much more simplified case, i.e., a steady 1D heat conduction in the three-layer media with the same sizes and thermal conductivities as in the heat exchanger shown in Fig. 2. Namely, the convection terms in Eq. (2) or Eq. (4) are given up ($U_1 = U_3 = 0$) and the left and right end-walls of the heat exchanger are assumed to be adiabatic. In addition, temperatures at the outer walls of the upper and lower flow passages are taken to be $T_u = 800$ K and $T_b = 300$ K, respectively, instead of the adiabatic boundary conditions used above. An exact analytical solution can be easily obtained for such a pure heat conduction problem. The analytical result of the heat flux is

$$q = (T_u - T_b)/[(\delta_1/k_f) + (\delta_2/k_s) + (\delta_3/k_f)]$$
$$= 2.27 \times 10^4 \text{ W/m}^2.$$

The temperature profile obtained from the exact solution has been plotted in Fig. 6 as the solid line. As expected, the temperature drop across the metallic partition plate is significantly less than that within the hot or cold fluid. Corresponding calculated results using the SIMPLE algorithm are also shown in Fig. 6 for Method A, Method B (or Approach 2) and Method C (or Approach 1), respectively. The computed results show that the temperature profiles obtained by Methods B and C are the same and are well consistent with the exact solution shown in Fig. 6 as the solid line. On the other hand, a different temperature profile is obtained by Method A as shown in Fig. 6 by the dash line. The result obtained using Method A is obviously incorrect.

Variable fluid specific heat may be encountered in many practical applications. For example, one order of magnitude variation in the specific-heat values is not unusual in a thermal plasma system with temperature difference as great as 10^4 K. For this case, Approach 2 is no longer applicable to treat Eqs. (4), (6) or (7). The reason is as follows. Referring to Fig. 1, if the specific heat of control volume 1 in the solid region is set to be equal to that of control volume 2 in the fluid region according to Approach 2, the specific heat of control volume 3 in the solid region must also be set to be equal to that of control volume 4 in the fluid region. Since now the fluid specific heat is temperature-dependent, the specific heat of control volume 2 usually is not equal to that of control volume 4 in the fluid region. Hence, the specific heat of control volume 1 will also be not equal to that of control volume 3 in the solid region. As a result, the heat flux continuity cannot be ensured at the interface between control volumes 1 and 3 in the solid region or even between control volumes 2 and 4 in the fluid region. On the other hand, Approach 1 is still valid for the variable fluid specific-heat case if an energy equation similar to Eq. (2) is used in the computation. Special attention should also be paid to the peculiarity of the energy equation. The pseudo-density method (Approach 1) will still be an effective method. Due to the complexity caused by the variable specific heat, the solution of the energy equation for such a case is more complicated than that for the constant specific-heat case. This problem has been discussed in some detail by Chen Xi and Li He-Ping (1999) in the modeling of a free-burning arc.

4. Conclusions

When conjugate heat transfer problems are solved using the SIMPLE-like algorithms in a unitary computational domain containing both the solid and fluid regions, one should pay close attention to ensure the physical realism of the computed results. In order to avoid any possible error, Approach 1 (i.e., the pseudo-density method) is recommended if the energy equation (2) is used, whereas Approach 2 (i.e., the pseudo-solid-specific-heat method) is recommended if energy equation (4), (6) or (7) is used.

Acknowledgements

This work was supported by the National Natural Science Foundation of China under grants nos. 59676011 and 59836220.

References

Chen Xi, Li He-Ping, 1999. Modeling of the high intensity free-burning arc: an improved approach. J. Phys. D, submitted.

Choi, C.J., Kim, S.J., 1996. Conjugate mixed convection in a channel: modified five percent deviation rule. Int. J. Heat Mass Trans. 39, 1223–1234.

Davalath, J., Bayazitoglou, Y., 1987. Forced convection cooling across rectangular blocks. ASME J. Heat Transfer 109, 321–328.

Gonzalez, J.J., Proulx, P., Boulos, M., 1993. Mathematical modeling of a free-burning arc in the presence of metal vapor. J. Appl. Phys. 74, 3065–3070.

Ju, Y., Chen, Z., 1996. Numerical simulation of natural convection in an enclosure with discrete protruding heaters. Numer. Heat Transfer A 30, 207–218.

Kim, D.M., Viskanta, R., 1985. Effect of wall heat conduction on natural convection heat transfer in a square enclosure. ASME J. Heat Transfer 107, 139–146.

Patankar, S.V., 1980. Numerical Heat Transfer and Fluid Flow. Hemisphere/McGraw-Hill, Washington, DC (Chapter 3).