AN ALGORITHM FOR SIMULATING THE ENERGY TRANSFER PROCESS IN A MOVING SOLID-FLUID MIXTURE

Maria Laura Martins Costa, Rubens Sampaio
Department of Mechanical Engineering - Pontifícia Universidade Católica - RJ
Rua Marquês de São Vicente,225 - 22452 - Rio de Janeiro - Brazil

Rogério M. Saldanha da Gama
Laboratório Nacional de Computação Científica (LNCC/CNPq)
Rua Lauro Müller,455 - 22290 - Rio de Janeiro - Brazil

ABSTRACT

In the present work an algotithm for a local simulation of the energy transfer phenomenon in a binary (solid-fluid) moving saturated mixture is proposed. An iterative procedure is used to simulate (by means of a Finite Difference approach) the heat transfer in a saturated flow (through a porous medium) between two parallel isothermal plates in which the fluid constituent's inlet temperature is the only boundary condition prescribed on x-direction. An exaustive number of tests have shown that the mentioned procedure (which is independent from initial estimates for both constituent's second order partial derivatives on x-direction) consists of an effective way to perform this simulation.

INTRODUCTION

The interest on flow through porous media, taking into account heat and/or mass transfer is growing significantly nowdays. Interactions between fluids and solids are present in many industrial processes. These fluids may be passed over packed beds of solid material, so that a large ratio of surface area to volume is obtained and phenomena such as heat and mass transfer or chemical reactions may occur. The main purpose of this work is to present a procedure which, despite its simplicity, is an effective way to perform a local simulation of the forced convection heat transfer process which occurs when a fluid flowing through a porous channel with realistic boundary conditions is considered.

While the well known classical (single continuum) energy transfer model [1] describes adequately the thermomechanical behaviour of materials such as steel, water, rubber or air, it is not so appropriate for a local description of the heat transfer process in a flow of a newtonian fluid through a porous medium. Such a description would require the solution (for the fluid) of both Navier-Stokes and Energy equations, in a domain defined by all active pores. Boundary conditions, such as no-slip condition and prescribed temperature (and/or heat fluxes), should be considered on all pore walls. The currently avaiable tools are not adequate to allow a simulation of so great degree of complexity.

In order to make possible a local description, the problem is regarded through a Continuum

Theory of Mixtures viewpoint [2]. A binary (solid-fluid) mixture is considered, in which the fluid, represented by the "fluid constituent", is assumed newtonian and incompressible, while the porous medium, represented by the "solid constituent", is assumed rigid, homogeneous, isotropic and at rest.

This model, which is supported by a Theory, with thermodynamical consistence, that generalizes the Classical Continuum Mechanics, allows a local description of the heat transfer phenomenon in a porous medium saturated by a fluid. The forced convection heating of a fluid which flows through a porous channel, bounded by two impermeable isothermal flat plates, is simulated with the mentioned model.

When the energy transfer between solid and fluid constituents is studied in a Continuum Theory of Mixtures viewpoint, the existence of two temperatures at each spatial point of the domain (the fluid and the solid constituents' temperatures) gives rise to the Energy Generation Function [3] (which provides the thermal interaction between both constituents of the mixture). The Continuum Theory of Mixtures demands each constituent to satisfy the balance equations, while a global balance equation must be satisfied by the mixture. The existence of the Energy Generation Function provides solid and fluid constituents'energy equations coupling.

When the forced convection heating of a fluid flowing through a porous channel is considered, in a two-dimensional geometry (as shown in Figure 1), a system of two second order partial differential equations on both x- and y-variables is to be solved. The characteristic nature of the energy equations allows this system to be solved with only one boundary condition on x-direction. From a mathematical viewpoint, this statment may sound absurd, but from a physical viewpoint, if both constituents' temperatures are prescribed on the channel's superior and inferior boundaries (y-direction) and the fluid inlet temperature is known, no aditional boundary condition seems necessary to determine both constituents' temperature fields. The use of aditional boundary conditions could, even, give rise to an unrealistic behaviour near the boundaries.

The main objective of the present work is to present a simple, but effective, procedure, capable of solving a system of second order partial differential equations, in two variables each, employing only five boundary conditions (instead of the usual seven or eight): four on y-direction and only one on x-direction.

In fact, an exaustive number of examples taken into consideration, has shown that the domain's interior is not affected by aditional boundary conditions on x-direction, which can lead to unrealistic situations on the boundaries.

Since both constituents' energy equations are elliptic, four boundary conditions should, in principle, be prescribed on x-direction. The fluid constituent's energy equation, however, because of its physical nature, is treated as a sequence of parabolic equations, suggesting that only one boundary condition (at the channel's entrance) would be expected to be necessary for its solution. Some tests, in which the fluid constituent inlet temperature was known and several values of the solid constituent partial heat flux (defined so as to be proportional to the solid and fluid constituents' temperatures difference) at the channel's entrance and exit were used, have confirmed the mentioned hypothesis.

The coupling of both energy equations suggested a step forward: to prescribe no condition for the solid constituent, either at the channel's entrance or at its exit. The verification of this hypothesis, which allows the phenomenon to be studied in a more (physically) realistic way, has motivated the present work.

MATHEMATICAL MODEL

Considering the mass and linear momentum balance equations for the fluid constituent (the porous a medium is assumed rigid and at rest and, therefore, doesn't need to satisfy such balance equa-

tions), assuming the two-dimensional geometry, presented in Figure 1, and one-dimensional steady-state flow, the following velocity profile is obtained [4]:

$$v_F = C\left(1 - \frac{\cosh\frac{y}{\sqrt{K\lambda}}}{\cosh\frac{H/2}{\sqrt{K\lambda}}}\right) \tag{1}$$

for $-H/2 \leq y \leq H/2$

in which C is a constant [4], H the channel's width, K the porous medium specific permeability, and λ a parameter depending on the porous matrix.

The energy balance [2] must be satisfied by each constituent of the mixture. Supposing steady-state conditions and zero heat generation for both constituents, it can be stated as:

$$\rho_i c_i (grad\ T_i)\cdot \mathbf{v}_i = -div\ \mathbf{q}_i + \psi_i \tag{2}$$

where $i \equiv S$ and $i \equiv F$ stand for the solid and the fluid constituents, respectively, ρ_i stands for the i-constituent density, T_i for its temperature, $\mathbf{q}_i$ and ψ_i represent, respectively its partial heat flux and energy generation function and, finally, c_i represents the specific heat of the i-constituent, regarded as a continuum.

Both energy equations are to be solved, in order to determine the two temperature fields (T_F and T_S),as the mixture theory viewpoint allows the existence of a different temperature for each constituent, at each spacial point.

Equation (2) requires some constitutive hypotheses. The partial heat fluxes for solid and fluid constituents ($\mathbf{q}_S$ and $\mathbf{q}_F$), according to the model proposed by Saldanha da Gama [3], are stated as:

$$\mathbf{q}_S = -\Lambda k_S(1-\varphi) grad\ T_S \qquad \mathbf{q}_F = -\Lambda k_F \varphi grad\ T_F \tag{3}$$

where Λ represents an always positive parameter which may depend on both the internal structure and the kinematics of the mixture, k_S and k_F are, respectivelly, the solid and the fluid constituents' thermal conductivity and φ the fluid fraction (coincident to the porosity, for saturated flows).

The total heat flux (per unit of time and area) for the mixture is given by the sum of $\mathbf{q}_S$ and $\mathbf{q}_F$.

The energy generation function, ψ, which is an internal contribution, represents the energy supply to a given constituent, arising from its (thermal) interaction with the other constituents of the mixture. The ψ function is zero at a given point only if all the constituents are at the same temperature at this point. According to Martins Costa [5], the energy generation function for solid and fluid constituents are given by:

$$\psi_S = -\psi_F = R(T_F - T_S) \tag{4}$$

where R is an always positive factor, which will be considered constant in this work.

Considering these constitutive hypotheses, the balance of energy for the fluid and the solid constituents can be written as:

$$\rho_F c_F (grad\ T_F)\cdot \mathbf{v}_F = \Lambda k_F \varphi \Delta(T_F) + R(T_S - T_F) \tag{5}$$

$$0 = \Lambda k_S(1-\varphi)\Delta(T_S) + R(T_F - T_S) \tag{6}$$

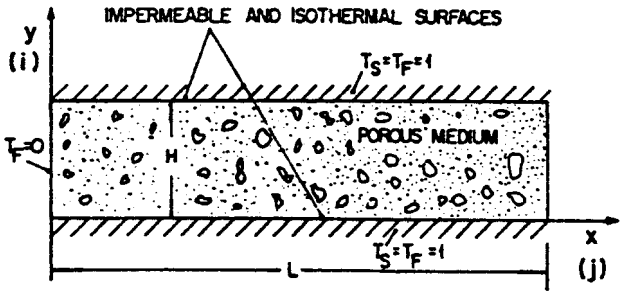

Figure 1 - Problem's Scheme

Since the fluid velocity is non zero only in the x-direction and considering the two-dimensional geometry of the problem, as shown in Figure 1, the balance of energy can be reduced to:

$$\alpha\frac{\partial T_F}{\partial x} = \left[\frac{\partial^2 T_F}{\partial x^2} + \frac{\partial^2 T_F}{\partial y^2}\right] + \beta(T_S - T_F) \tag{7}$$

$$0 = \frac{\partial^2 T_S}{\partial x^2} + \frac{\partial^2 T_S}{\partial y^2} + \gamma(T_F - T_S) \tag{8}$$

where:

$$\alpha = \frac{\rho_F c_F v_F}{\Lambda k_F \varphi} \qquad \beta = \frac{R}{\Lambda k_F \varphi} \qquad \gamma = \frac{R}{\Lambda k_S (1-\varphi)} \tag{9}$$

satisfying the following boundary conditions:

$$T_F(0,y) = 0 \ , \qquad T_F(x,0) = T_S(x,0) = T_F(x,H) = T_S(x,H) = 1 \tag{10}$$

NUMERICAL METHOD

The problem consists of a system of two second order equations, on both x and y-variables, subjected to four boundary conditions on y-direction and to only one boundary condition on x-direction. From a mathematical viewpoint, a problem of this kind, composed of two elliptic equations, on both x- and y-variables, even if physically realistic, could give rise to an infinite number of solutions. However, a great number of tested situations has shown that aditional boundary conditions on x-direction have no influence on both solid and fluid constituents' bulk temperatures.

An iterative procedure is used, so that two second order equations on x-variable can be solved with the help of a single boundary condition on x-direction: the fluid constituent's inlet temperature. The problem is treated as a sucession of modified problems in which the second order derivatives on x-direction, for both constituents, are treated as previously known fields, that is: the fluid constituent's energy equation is treated as a sequence of parabolic problems on x-variable, while the solid constituent's energy equation can be considered as a sequence of elliptic problems on y-variable. This procedure can be summarized in the following way: at the first iteration initial values are estimated for these two fields, approximations for them being calculated after the solution of the modified problem. These approximations are used as the second order derivatives' values in

the next iteration, instead of the initial values, and new approximations for the derivatives are calculated after the modified problem's solution. In short, the process consists of calculating the n-th iteration, using, for the diffusive terms on x-direction, approximations calculated in the n-1 iteration. This process is repeated until further iterations cease to produce significative alteration on temperature values. The original system of equations is modified to:

$$\left[\alpha\frac{\partial T_F}{\partial x} - \frac{\partial^2 T_F}{\partial y^2} - \beta(T_S - T_F)\right]^l = \left[\frac{\partial^2 T_F}{\partial x^2}\right]^{l-1} \tag{11}$$

$$\left[-\frac{\partial^2 T_S}{\partial y^2} - \gamma(T_F - T_S)\right]^l = \left[\frac{\partial^2 T_S}{\partial x^2}\right]^{l-1} \tag{12}$$

where the derivatives $\frac{\partial^2 T_F}{\partial x^2}$ and $\frac{\partial^2 T_S}{\partial x^2}$ are calculated from a previous iteration.

Since no analitical solution to the system of equations describing the problem is known, numerical approximations to its solution are searched with the help of a finite difference approach [6]. For the diffusive terms, a central finite difference scheme discretization was used, while an "Upwind" scheme [6] was employed in the convective term discretization.

As the temperature coeficients' matrix (associated to the modified system of equations) is a sparse one, a grid description, in which each constituent's temperature possesses two indexes, according to its position on the grid, is used. Each iteration l is then solved with the help of the Gauss-Seidel method, according to the following discretized system:

$$\eta[T_F^{k+1}(i,j)]^l = \delta\{[T_F^{k+1}(i-1,j)]^l + [T_F^k(i+1,j)]^l\} +$$
$$+\nu[T_F^{k+1}(i,j-1)]^l + \beta[T_S^k(i,j)]^l + F\left[\frac{\partial^2 T_F}{\partial x^2}\right]^{l-1} \tag{13}$$

$$\mu[T_S^{k+1}(i,j)]^l = \delta\{[T_S^{k+1}(i-1,j)]^l + [T_S^k(i+1,j)]^l\} +$$
$$+\gamma[T_F^{k+1}(i,j)]^l + F\left[\frac{\partial^2 T_S}{\partial x^2}\right]^{l-1} \tag{14}$$

In (13) and (14), $2 \le i \le Nx$ and $2 \le j \le Ny$, in which Nx and Ny are the number of divisions on x- and y-directions, respectivelly. The approximations for both constituents' second order partial derivative, calculated from a previous (l-1) iteration, are given by the following discretized equations:

$$F\left[\frac{\partial^2 T_F}{\partial x^2}\right]^l \approx \frac{T_F^{l-1}(i,j+1) - 2T_F^{l-1}(i,j) + T_F^{l-1}(i,j-1)}{(\Delta x)^2} \tag{15}$$

$$F\left[\frac{\partial^2 T_S}{\partial x^2}\right]^l \approx \frac{T_S^{l-1}(i,j+1) - 2T_S^{l-1}(i,j) + T_S^{l-1}(i,j-1)}{(\Delta x)^2} \tag{16}$$

where Δ x is the mesh size on x-direction, l represents the global iteration, k the Gauss-Seidel iteration and:

$$\eta = \frac{\alpha}{(\delta x)} + \frac{2}{(\delta y)^2} + \beta \qquad \mu = \frac{2}{(\delta y)^2} + \gamma \qquad \delta = \frac{1}{(\delta y)^2} \qquad \nu = \frac{\alpha}{(\delta x)} \tag{17}$$

Equations (13) and (14) represent the modified problem in a very simple way, which allows an effective storage scheme, with memory reutilization.

The iterative procedure, represented by equations (13) to (17), was repeated to a great variety of initial estimate values of the second order partial derivative on x-direction, ranging from -10^4 to $+10^4$. In all these cases the same results for the solid and fluid constituents' temperature fields were obtained, although the velocity of convergence showed a slight variation. In some of the tested cases, not only the derivatives' initial estimates, but also the factor R (which causes both solid and fluid constituents' energy equations coupling) was varied. Convergence to a same set of temperature fields (according to the value of R) was observed for all tested cases. This is a strong argument for the validity of the exposed procedure. Another meaningful argument is that two differenrt sets of similar problems, where the complete energy balance equations are considered (one without the described iterative procedure to calculate the second order derivative approximations and the other using it only for the fluid constituent) together with different boundary conditions, were simulated and compared to the problem in question. In the first type of problem, several fluid constituent's outlet temperatures (ranging from 0 to 1) were prescribed, while zero heat flux was prescribed for the solid constituent both at the channel's entrance and exit. For the second type of problem, a similar iterative procedure was used only for the fluid constituent, and several values of the solid constituent's heat flux were also considered, both at inlet and outlet, by varying a heat transfer coeficient, h, analogous to the one usually employed in the classical Newton's law of cooling, in equations:

$$\mathbf{A}k_S(1-\varphi)\frac{\partial T_S}{\partial x}(0,y) = h[T_S(0,y) - T_F(0,y)] \tag{18}$$

$$-\mathbf{A}k_S(1-\varphi)\frac{\partial T_S}{\partial x}(L,y) = h[T_S(L,y) - T_F(L,y)]$$

In all these cases no alteration on both constituents' temperature profile, except for the channel's entrance and/or exit, is observed.

The above stated arguments seem sufficient to validate the numerical procedure employed in the present work.

Convergence criterium for both Gauss-Seidel and global iterations was:

$$\max_{(i,j)}\{|T_F^m(i,j) - T_F^{m-1}(i,j)|, |T_S^m(i,j) - T_S^{m-1}(i,j)|\} \leq 10^{-6} \tag{19}$$

where $1 \leq i \leq (Nx+1)$, $1 \leq j \leq (Ny+1)$, $m \equiv l$ represent the global iterations and $m \equiv k$ represent the Gauss Seidel iterations.

A very quick convergence of the l-iterations was observed, four global iterations being sufficient for the worst case. The velocity of convergence of the intermediate iterations (Gauss-Seidel method) varied also, according to the second order partial derivatives' initial values.

RESULTS

In this section some results, considering a long porous channel (with 120 lenght and 1 heigh) divided into a 13x13 grid as default, are presented. In Figures 2 and 3 this default problem is compared, respectivelly to a problem where zero heat flux is prescribed for the solid constituent on both channel's edges, while several values for the fluid constituent's outlet temperature are prescribed, and to another problem where two different values of solid constituent's heat flux are considered (on both channel's edges) while no boundary condition is imposed to the fluid constituent at the channel's exit.

Figures 4 and 5 plot both constituents' temperature (at the channel's central point) for different mesh sizes and its percentual difference, related to the most refined grid considered, respectivelly.

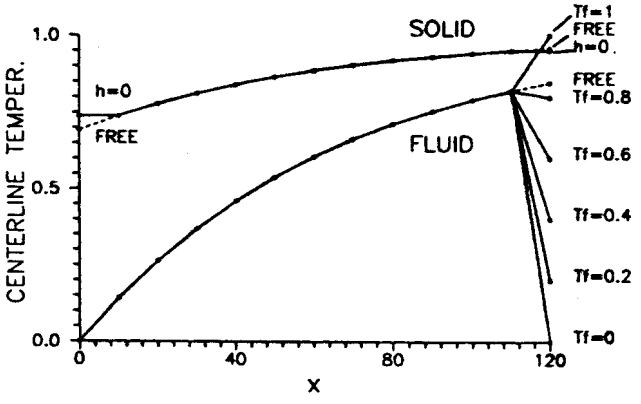

Figure 2 - Centerline Temperatures vs x
(varying T_F at the channel's exit)

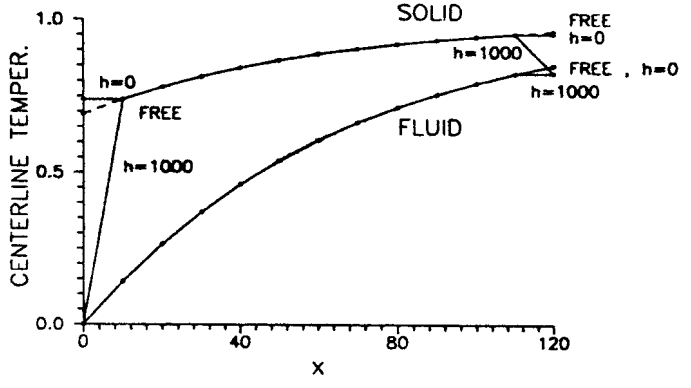

Figure 3 - Centerline Temperatures vs x
(varying q_S both at inlet and outlet)

Figure 6 compares the temperature behaviour when mesh size is reduced 50 % on x-direction. Finally, in Figure 7, the channel's length is considered five times greater than the default value.

Figure 2 shows both constituents' centerline temperatures versus the x-variable in two different cases. The first one (represented by the dashed lines, for both constituent's curves) shows the problem, whose simulation originated the present work: no boundary conditions are prescribed either for the fluid constituent at the channel's exit or for the solid constituent both at the channel's entrance and exit, as stated in equation (10). The continuous curves correspond to the second case, where zero heat flux was prescribed for the solid constituent, both at inlet and outlet. Six different fluid constituent's curves correspond to the described solid constituent's curve, according to the prescribed outlet fluid constituent's temperatures. Several values were considered for this temperature, varying from 0 (the fluid constituent prescribed inlet temperature) to 1 (the impermeable

isothermal surfaces prescribed temperature). This second problem, in which four boundary conditions were prescribed on x-direction, shows an artificial behaviour, both at the channel's entrance and exit. Except for these values, complete agreement can be verified between the temperature fields, for both cases taken into consideration.

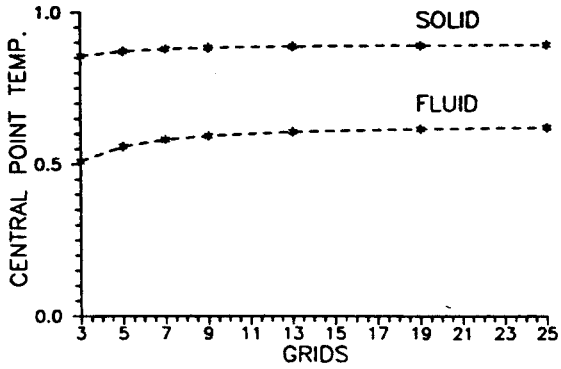

Figure 4 - Central Point Temperatures (for different mesh sizes)

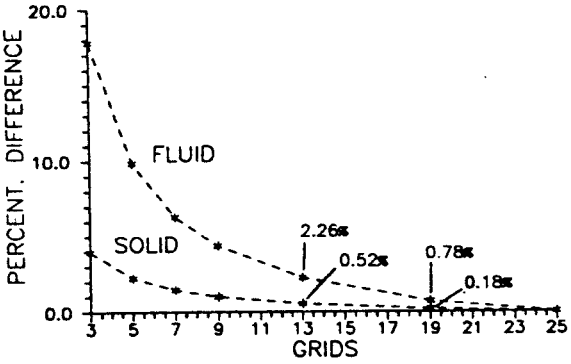

Figure 5 - Percentual Difference among Central point Temperature (for different Mesh Sizes, related to the most refined grid)

Figure 3 shows a comparison between two different problems, the first one, represented by the dashed lines, as in Figure (2), is the one described by equations (7) to (10), while in the other case several heat fluxes are considered, by varying the heat transfer coeficient h in equation (18), for the solid constituent, both at the channel's entrance and exit, while no boundary condition for the fluid constituent at the outlet is prescribed. The latter problem, represented by the continuous lines, is solved by means of an iterative scheme, in which the elliptic fluid constituent's problem is solved

as a sequence of parabolic problems. This scheme is similar to the one described in the present work, but only the fluid constituent's second order partial derivative on x-direction is treated as a known field. It is remarkable that no variation on the fluid constituent's temperature curve is observed, when no boundary condition or zero heat flux was prescribed for the solid constituent. A very slight difference between these mentioned cases is observed at the channel's exit, while a more significative difference can be observed at its entrance. The use of a heat transfer coeficient so great as $h = 1000$ is almost equivalent to prescribe both solid and fluid constituents' temperatures with the same value. As a consequence, a value very close to zero is observed at the channel's entrance for the solid constituent's temperature, while apparently the same temperature values for both constituents can be observed at the channel's exit. This problem was considered for several values of h, between 0 and 1000, and, except for the channel's entrance and exit, no difference on both constituents' temperature fields is observed, as occurred on the case shown in Figure 2.

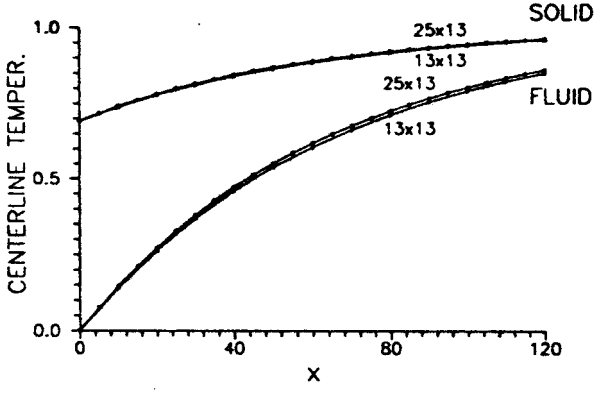

Figure 6 - Centerline Temperatures vs x (for 25x13 and 13x13 grids)

Figure 4 shows both constituents' temperatures at a point located at the centerline's center, for different meshes, from a 3x3 mesh to a 25x25 one. The percentual difference among the latter mesh and the remaining ones, at the central point, is plotted for both constituents' temperatures in Figure 5. Examining Figures 4 and 5 together, it can be noticed that the 13x13 grid, used for the majoriry of the results presented in this work, shows a reasonable agreement to the most refined one used: the 25x25 grid, for both constituents' temperatures.

Figure 6 shows both constituents' centerline temperatures for two different mesh sizes: 25x13and 13x13. A very slight difference is observed for the fluid constituent's temperature, while almost no difference can be noticed for the solid constituent's, as the grid is refined on x-direction. The difference between these two curves is more accentuated near the channel's exit. Comparing both constituents' temperature curves at a section $x = 110$, near the channel's exit, it can be seen that, even near the channel's exit, only a little difference between these curves is observed.

Figure 7 represents both constituents' centerline temperatures versus x-variable, for $L = 600$, five times greater than $L = 120$ (the length considered in Figures 2 to 6). It can be noticed that both constituents'centerline temperatures are almost coincident at the second half of the channel. When this Figure is compared to any one in which centerline temperatures are plotted versus x-variable,

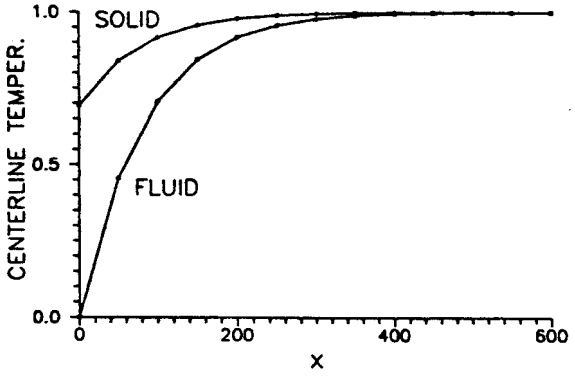

Figure 7 - Centerline Temperatures vs x
(for $L = 600$)

for $L = 120$, the concept of developed temperature fields, after a certain value of the channel's length, becomes a natural expectation.

FINAL REMARKS

When a problem like the one stated in equations (7) and (8) is simulated, usually a total of eight boundary conditions is required. However, the practical situation considered in the present work becomes unrealistic if all the usual boundary conditions are prescribed.

This work presents an algorithm which allows the local simulation of the energy transfer process in a saturated flow throgh a rigid porous medium, using a Mixtures Theory viewpoint, in which a system of two elliptic equations on both x- and y-variables are solved with only one boundary condition on x-direction: the fluid constituent's inlet temperature. Aditional data like the fluid constituent's temperature or heat transfer and the solid constituent's inlet and outlet temperature and/or heat transfer, which are not avaiable in practical problems, need not to be known.

REFERENCES

1. Slattery, J. C., *Momentum, Energy and Mass Transfer in Continua*, 2nd edn, Robert E. Krieger Publishing Co., New York, 1981.
2. Atkin, R.J. and Craine, R.E., *Continuum Theories of Mixtures. Basic Theory and Historical Development*, Quart. J. Mech. appl. Math., Vol. 29, 209-244 (1976).
3. Saldanha da Gama, R.M., *A Model for Heat Transfer in Rigid Continuous Mixtures*, Int. Comm. Heat Mass Transfer, Vol. 16, 123-132 (1989).
4. Martins Costa, M. L., Sampaio, R. and Saldanha da Gama, R. M., *Modelling and Simulation of the Energy Transfer Phenomenon in a Continuous Solid-Fluid Mixture* (In Portuguese), Proceedings of the X Iberic-Latin-American Congress on Computational Methods in Engineering, Vol. 4, D1-D11, Porto, Portugal (1989).
5. Martins Costa, M. L., *Modelling and Simulation of the Energy Transfer in Saturated Porous Media by using Theory of Mixtures*, (In Portuguese), DSc. Thesis, Pontifícia Universidade Católica do Rio de Janeiro, Brazil, 1981.
6. Euvrard, D., *Résolution Numérique des Équations aux Derivées Partielles*, 1st edn, Masson, Paris, 1987.