X CONGRESSO BRASILEIRO DE ENGENHARIA MECÂNICA, RIO DE JANEIRO, RJ (DEZEMBRO, 1989)
10th BRAZILIAN CONGRESS OF MECHANICAL ENGINEERING

THE USE OF CO-LOCATED VARIABLES IN THE SOLUTION OF SUPERSONIC FLOWS

PEM
PROGRAMA DE ENGENHARIA MECÂNICA
COPPE/UFRJ

C. H. MARCHI, C. R. MALISKA & A. L. BORTOLI
Department of Mechanical Engineering
Federal University of Santa Catarina
P.O.Box, 476 - 88049 - Florianópolis - SC - Brazil

ABSTRACT

The solution of incompressible flows using control volume methods normaly employ the staggered grid arrangement in order to have a tight coupling between pressure and velocity. This procedure introduces significant difficulties in the computer code implementation due to the existence of different control volume for each variable. This paper presents the numerical solution of compressible and incompressible flows using a general methodology for all speed flows, written for co-located variables. Incompressible and supersonic flows are solved and the results compared with the ones obtained using staggered grids in terms of convergence behaviours and CPU effort.

INTRODUCTION

The numerical methods designed to solve supersonic flow problems employ, in general, the state equation for finding pressure. In this case density is considered the dependent variable in the mass conservation equation. When the incompressible limit is reached, that is, for low Mach numbers, these methods are no longer suitable.

Very recently, extensions of the metodologies employed for incompressible flows have been applied, with success, in the solution of compressible fluid flow problems. These methods form an equation for pressure, replacing, in the mass conservation equation, density by a linearized form of the state equation and velocity components by their respective momentum equations. The drawback of these methods is that they require the use of staggered variables in order to provide the adequate coupling between pressure and velocity/density. As a consequence of the staggered arrangement the computer code implementation becomes cumbersome, specially if variable grid spacing is used in three dimensions, because the different control volume locations and the corresponding metric storaging.

The alternative to this problem is to keep all variables stored at the same point, that is, all of them share the same elemental control volume. The use of co-located variables simplifies considerably the coefficients calculation and storaging, and geometrical data storaging. The difficulty associated with the use of co-located variables is the poor coupling it provides between pressure and velocity/density. This difficulty can be removed taking care in numerically approximating the pressure gradients.

Successful aplications of finite volume methods using nonorthogonal grids with co-located variables can be seen in [1] and [2]. With the increase in the use of nonorthogonal grids, the interest in using co-located variables is growing [3] [4]. They, however, deal only with incompressible fluid flows and only very few numerical experiments have been done employing such methods up to now.

This paper advances the application of co-located variables for viscous compressible flows employing a simultaneous correction in velocity and density to deal with the pressure-velocity/density coupling [5] [6]. Preliminary results are reported for the developing flow (incompressible) between paralel plates, with the only purpose of code checking, and for the viscous compressible flow over an obstacle with Mach number equal to 2.0. The results obtained demonstrate that co-located variables are a viable route to follow for the development of more compact and easy to implement computer codes written for nonorthogonal grids.

The paper also contributes to the knowledge of the behaviour of co-located methods for compressible flows. This is of major importance since the variables location in the grid is intimately connected with the strength of the pressure-velocity/density coupling.

GOVERNING EQUATIONS AND APPROXIMATE EQUATIONS

The governing equations for laminar compressible flows with constant physical properties can be written for a general scalar ϕ as

$$\frac{\partial}{\partial t}(\rho\phi) + \frac{\partial}{\partial x}(\rho u\phi) + \frac{\partial}{\partial y}(\rho v\phi) = \Gamma^{\phi}\frac{\partial}{\partial x}\left(\frac{\partial \phi}{\partial x}\right) + \Gamma^{\phi}\frac{\partial}{\partial y}\left(\frac{\partial \phi}{\partial y}\right) + S^{\phi} - P^{\phi} \qquad (1)$$

where the Γ^{ϕ}, S^{ϕ} and P^{ϕ} terms can be found in Table 1.

Table 1. Expressions for Γ^{ϕ}, S^{ϕ} and P^{ϕ}.

ϕ	Γ^{ϕ}	S^{ϕ}	P^{ϕ}
1	0	0	0
u	μ	$\frac{\mu}{3}\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)$	$\frac{\partial p}{\partial x}$
v	μ	$\frac{\mu}{3}\frac{\partial}{\partial y}\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right)$	$\frac{\partial p}{\partial y}$
T	$\frac{k}{c_p}$	$\left(\frac{\partial p}{\partial t} + \frac{\partial}{\partial x}(up) + \frac{\partial}{\partial y}(vp) - p\frac{\partial u}{\partial x} - p\frac{\partial v}{\partial y}\right)/c_p$	0

To integrate the Eq.(1), to obtain the algebric equations, the variable location in the grid must be decided. In this paper one is interested in analysing the performance of the co-located variables method. As can be seen in Fig. 1, u, v, p, ρ and T are all located at the point P.

There are two main questions to be answered at this point when using this type of variable location. First, when integration of the mass conservation equation is performed there will be the need of calculating velocities at the control volume interfaces, since velocity components are not stored at those locations. How to compute these velocities? Also, one must decide how to calculate the pressure gradient which will enter the momentum equations, such that a tight coupling between pressure and velocity is assured. In fact the answer to these two questions, such that mass is conserved, togheter with an adequate pressure-velocity coupling characteristic, enable the use of co-located variables for the solution of fluid flow and heat transfer problems.

To illustrate the integration of the Eq.(1), the integration of the second term in the left hand side, over the elemental control volume shown in Fig. 1 is performed. This gives

$$\int_{t}^{t+\Delta t}\int_{y_s}^{y_n}\int_{x_w}^{x_e}\frac{\partial}{\partial x}(\rho u\phi)dxdydt = (\rho_e\overline{u}_e - \rho_w\overline{u}_w)\Delta y\Delta t$$

It is recalled that $\overline{u}_e$, $\overline{u}_w$, $\overline{v}_n$ and $\overline{v}_s$ are not known and must be evaluated as a function of the velocity components stored at the center of the control volumes. Ways to find these velocities will be discussed in this paper.

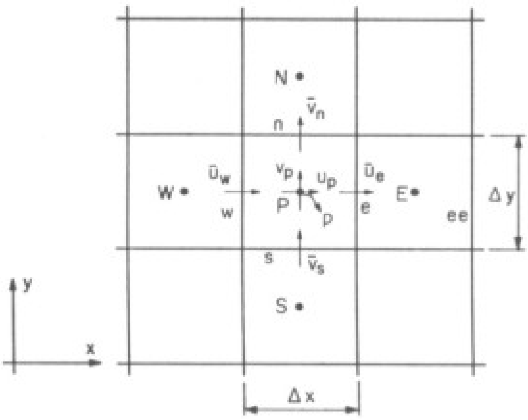

Fig. 1 - Location of the variables on the grid.

Integration of Eq.(1) over time and over the elemental control volume shown in Fig. 1, gives

$$a_{PP}\phi_P = \Sigma(a_{nb}\phi_{NB})_P + M_P^o\phi_P^o/\Delta t + L[S^\phi]_P\Delta V - L[p^\phi]_P\Delta V \quad (2)$$

Again it is recalled that when ϕ is equal to u or v, the pressure needed in the momentum equations will be p_e and p_w, and p_n and p_s, respectivelly.

EVALUATION OF $\overline{u}$, $\overline{v}$ AND p AT INTERFACES

Two schemes to evaluate $\overline{u}$ and $\overline{v}$ at the interfaces are used. Both of them calculate $\overline{u}_e$ as a function of u_P and u_E. The two velocities are given by

$$a_{PP}u_P = \left(\Sigma(a_{nb}u_{NB})_P + M_P^o u_P^o/\Delta t + L[S^u]_P\Delta V\right) - \Delta V(p_e - p_w)/\Delta x \quad (3)$$

$$a_{PE}u_E = \left(\Sigma(a_{nb}u_{NB})_E + M_E^o u_E^o/\Delta t + L[S^u]_E\Delta V\right) - \Delta V(p_{ee} - p_e)/\Delta x \quad (4)$$

or

$$a_{PP}u_P = \hat{a}_{PP}\hat{u}_P - \Delta V(p_e - p_w)/\Delta x \quad (5)$$

$$a_{PE}u_E = \hat{a}_{PE}\hat{u}_E - \Delta V(p_{ee} - p_e)/\Delta x \quad (6)$$

where a_{PP} is the central coefficient at volume P and a_{PE} is the central coefficient at volume E.

The pressure gradient which drives the $\overline{u}_e$ velocity will be taken as $(p_E - p_P)/\Delta x$, creating the consistency between the pressure gradient and the velocities which takes part in the mass conservation equation. Therefore, to form the equations for the interface velocities the pressure gradients in Eq.(5) and (6) do not enter in the averaging process.

Scheme A. For this case the $\overline{u}_e$ is calculated by

$$\overline{u}_e = \frac{1}{2}\frac{\left(\hat{a}_{PP}\hat{u}_P + \hat{a}_{PE}\hat{u}_E\right)}{(a_{PP}+a_{PE})/2} - \frac{\Delta V}{\Delta x}\frac{\left(p_E - p_P\right)}{(a_{PP}+a_{PE})/2} \quad (7)$$

As can be seen, not considering the pressure gradient the $\overline{u}_e$, as calculated above, is an average of the two neighbouring velocities weighted by their central coefficients and source terms. This is the procedure adopted in this paper.

Scheme B. This scheme is the one used by Peric [4] in the calculation of incompressible fluid flow problems. The $\overline{u}_e$ is evaluated as

$$\overline{u}_e = \frac{1}{2}\left[\frac{\hat{a}_{PP}\hat{u}_P}{a_{PP}} + \frac{\hat{a}_{PE}\hat{u}_E}{a_{PE}} - \frac{\Delta V}{\Delta x}\left(\frac{1}{a_{PP}} + \frac{1}{a_{PE}}\right)(p_E - p_P)\right] \quad (8)$$

Inspection of Eq.(8) reveals that this averaging process is a linear interpolation, without considering the pressure gradient, between the neighbouring velocities. It is demonstrated in [7] that without a proper underrelaxation parameter this scheme leads to a different steady solutions for different time steps. Calculations carried out during this work using this scheme revealled the same behaviour for the compressible calculations. In contrast, the scheme proposed in this work converges for a unique steady state solution independent of the time step used.

For completness the $\overline{u}_e$ velocity used in this work is rewritten here

$$\overline{a}_{Pe}\overline{u}_e = (\hat{a}_{PP}\hat{u}_P + \hat{a}_{PE}\hat{u}_E)/2 - \Delta V(p_E - p_P)/\Delta x \quad (9)$$

where $\overline{a}_{Pe}$ is the average of the central coefficients for u_P and u_E. Similar expressions can be written for $\overline{u}_w$, $\overline{v}_n$ and $\overline{v}_s$. These velocity components will enter the mass conservation equation.

PRESSURE-VELOCITY/DENSITY COUPLING

The mass conservation equation is recovered putting $\phi = 1$ and Γ^ϕ, S^ϕ and p^ϕ equal to zero in Eq. (1). The equation is

$$(M_P - M_P^o)/\Delta t + \dot{M}_e - \dot{M}_w + \dot{M}_n - \dot{M}_s = 0 \quad (10)$$

where, to ilustrate the mass flux linearization, the $\dot{M}_e$ term is taken as example

$$\dot{M}_e = \rho_e\overline{u}_e\Delta y = \rho_e\overline{u}_e^*\Delta y + \rho_e^*\overline{u}_e\Delta y - \rho_e^*\overline{u}_e^*\Delta y \quad (11)$$

where the star means the value of the best estimate value of ρ and $\overline{u}_e$. This mass flux linearization permits fluid flow problems to be solved for all speeds, that is, ranging from incompressible to high Mach number flows. In this procedure velocity and density are corrected using a correction pressure field. Expressions for the velocity corrections follows the SIMPLEC [8] procedure and are given by, for the $\overline{u}_e$ velocity,

$$\overline{u}_e = \overline{u}_e^* - \overline{d}_P^u(p'_E - p'_P) \qquad (12)$$

Similar expressions are found for $\overline{u}_w$, $\overline{v}_n$ and $\overline{v}_s$. The correction equation for ρ is found from the state equation as

$$\rho_P = \rho_P^* + C_P^\rho p'_P \qquad (13)$$

Substituting ρ, $\overline{u}$ and $\overline{v}$ by their respectives correction equations in the mass conservation equation, using the mass flux linearization equation given by Eq.(11), one obtains an equation for the pressure correction, as

$$a_P^P p'_P = a_e^P p'_E + a_w^P p'_W + a_n^P p'_N + a_s^P p'_S + b^P \qquad (14)$$

A complete description of the method used to treat the pressure-velocity/density coupling can be found in [6]. The same procedure described in [6] applies here, substituting the velocity components which enter mass conservation by the expressions given by Eq. (5) and (6). Briefly, a simplified solution procedure can be summarized as follows:

1. Estimate a u, v, T and p fields.
2. Solve for u^* using Eq. (3) and similar equation for v^*. The pressure gradiente for these equations are calculated using a linear interpolation for pressure.
3. Calculate ρ^* using the state equation.
4. Calculate $\vec{W}^*$ and solve Eq.(14) to obtain p'.
5. Correct u^*, v^* and ρ^* using Eq. (12) and (13).
6. Solve for T.
7. Back to step 2 and iterate until convergence.

NUMERICAL RESULTS

As mentioned, two test problems are solved using the co-located and the staggered arrangements. The objectives of the tests are two-fold. Firstly, it is to compare the results obtained with the co-located against the staggered arrangement, based on the quality of the results and CPU effort. Secondly, it is to observe the convergence characteristics of the co-located arrangement when the time step to advance the solution is changed. The first test problem is the laminar incompressible flow in the entrance region of two paralel plates. Since the methodology used can handle any flow regime, the Mach number was taken equal to 5x10-5 in order to have a truly incompressible flow. Fig. 2 shows the velocity profile for different axial stations, calculated with 22x18 volumes. The results are exactly the same as using the staggered arrangements for both schems A and B of evaluating the interface velocities. The CPU effort is also the same for both arrangements. The first important finding is the fact that the way the interface velocity is evaluated does not influence the results and the CPU effort to obtain the solution when incompressible flow problems are solved.

The second test problem is the supersonic flow over an obstacle as shown in Fig. 3, where the boundary conditions are also presented. The Mach number used was equal to 2.0. Fig. 4 shows the constant pressure lines for both the co-located and staggered arrangements, calculated with a (22x18) grid. It is seen that behind the step the agreement is not good. The constant pressure lines were chosen to be plotted because the other fields present less dicrepancies. Fig. 5 depicts the same fields but at this time obtained with a (44x36) grid. It can be seen now that the results agree very well. It is important to point out that the code which uses staggered grids was already tested previously [6].

As final results Fig. 6 and 7 present the convergence behaviour of both methodologies. In the results which follows, scheme A was used in the co-located metodology, since scheme B produced different steady state solutions for different time steps.

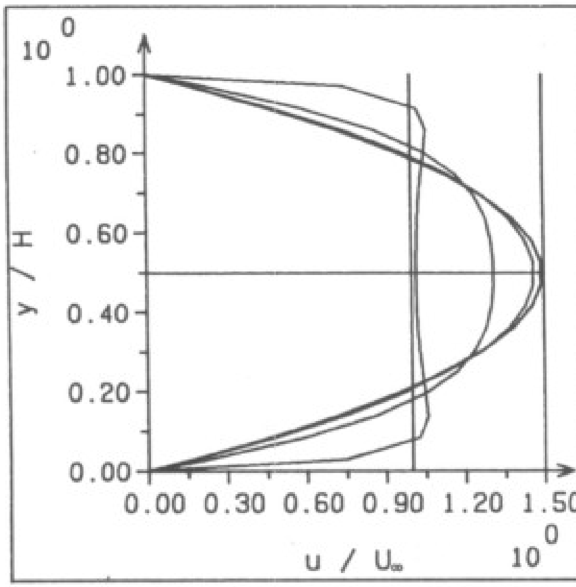

Fig. 2 - Velocity profiles for different axial stations to Mach 5x10-5.

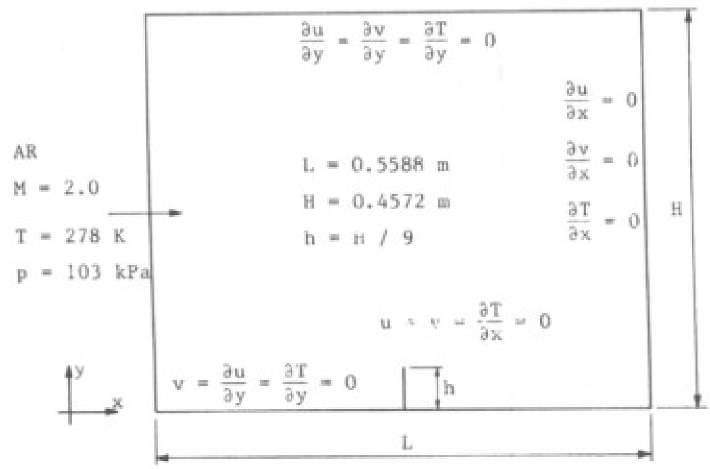

Fig. 3 - Geometry and boundary conditions.

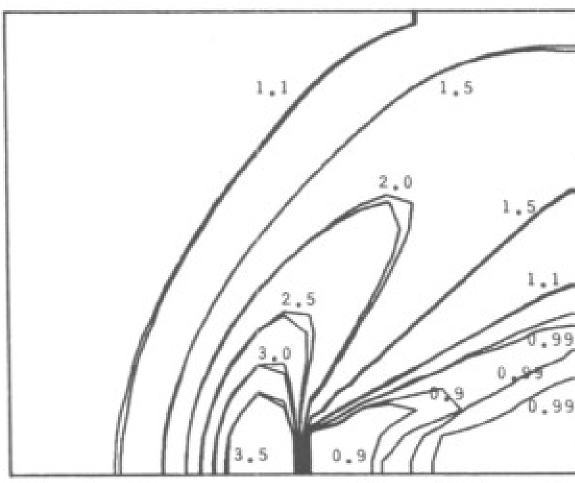

Fig. 4 - Constant pressure lines. Co-located and staggered arrangements. 22x18 grid.

Fig. 6 shows the convergence behaviour using a (22x18) grid, where it can be observed that the optimum time step to advance the solution, using co-located variables, is located in a narrow range if compared with the staggered approach. Also, the CPU effort to obtain the solution to a same level of tolerance, is twice as much for the co-located variables. Increasing the grid resolution (44x36) the performance of both methodologies are comparable. It can be seen that the range of optimum time step is almost the same with equal CPU effort to obtain the solution to a specified level of tolerance.

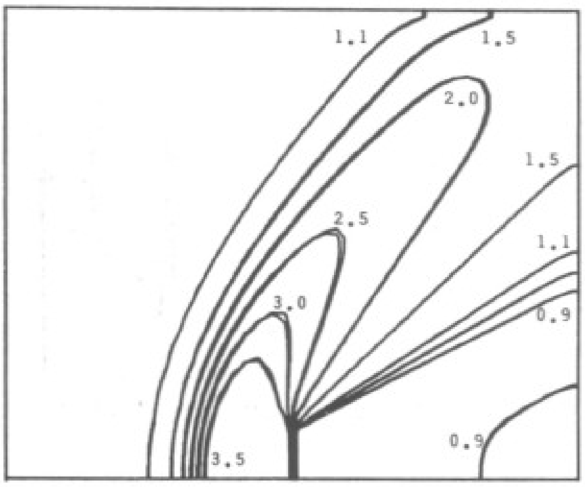

Fig. 5 - Constant pressure lines. Co-located and staggered arrangements. 44x36 grid.

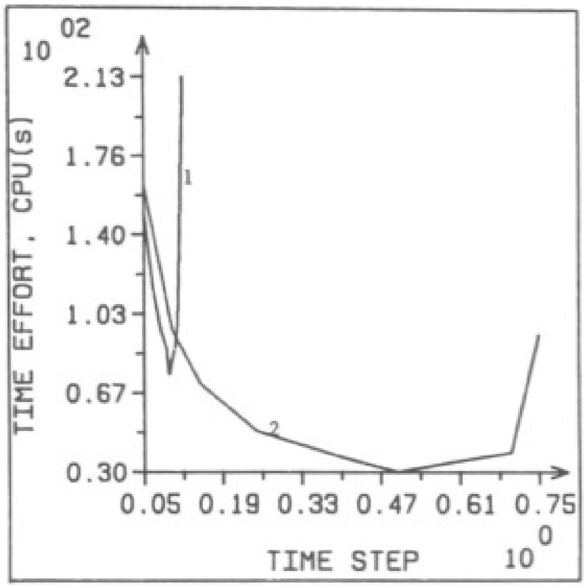

Fig. 6 - Convergence behaviour for the co-located (1) and staggered (2) arrangements. 22x18 grid.

CONCLUSIONS

The preliminary results obtained with the solution of two dimensional fluid flow problems using co-located variables demonstrated that the pressure-velocity (incompressible flows) or the pressure-velocity/density (compressible flows) coupling can be adequately taken into account without the compulsory use of staggered grids. The great advantages are the simplicity it promotes in the code implementation, as well as a significant reduction in storage requirements, since the convection and diffusion fluxes need to be calculated for only one control volume. If non uniform grids are used, as is the case when general discretizations are employed, the use of co-located variables reduces significantly the geometric information storaging. Results using nonorthogonal coordinates with co-located variables for supersonic flows, over arbitrary bodies are now being obtained and will be published in a future work.

The authors opinion is that extension of general methodologies for three dimensional flows must employ co-located variables in order to obtain an easy to implement computer code.

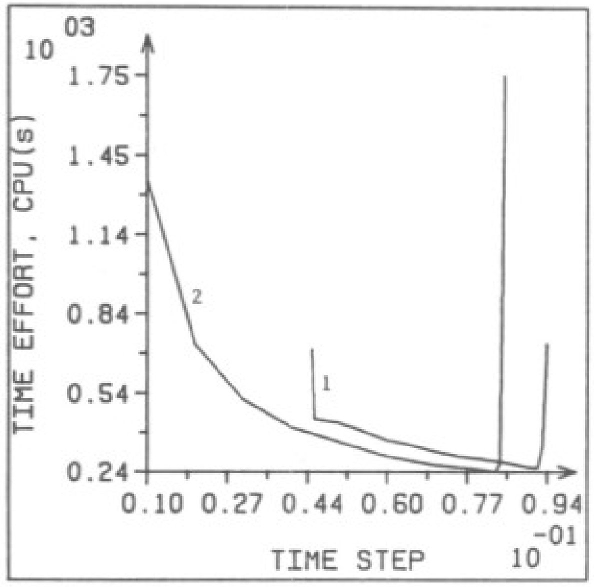

Fig. 7 - Convergence behaviour for the co-located (1) and staggered (2) arrangements. 44x36 grid.

REFERENCES

[1] HSU, C. "A Curvilinear-Coordinate Method for Momentum, Heat and Mass Transfer in Domains of Irregular Geometry". PhD Thesis, University of Minnesota (1981).

[2] RHIE, C.M. "A Numerical Study of the Flow Past an Isolated Airfoil with Separation". PhD Thesis, University of Illinois, Urbana-Champaign (1981).

[3] RHIE,C.M. & CHOW,W.L. "A Numerical Study of the Turbulent Flow Past an Isolated Airfoil with Trailing Edge Separation". AIAA-Paper-82-0998 (1982).

[4] PERIC,M., KESSLER,R. & SCHEUERER,G. "Comparison of Finite-Volume Numerical Methods with Staggered and Colocated Grids". Submitted to Computers and Fluids, 1987. 56 p.

[5] VAN DOORMAAL,J.P., RAITHBY,G.D. & MCDONALD,B.H. "The Segregated Approach to Predicting Viscous Compressible Flows". ASME paper no. 86-GT-196, presented at the Intl. Gas Turbine Conference and Exhbit, Dusseldorf, West Germany, June 8-12, 1986.

[6] MALISKA, C.R. & SILVA, A.F.C. "Desenvolvimento de Códigos Computacionais para Solução de Problemas de Escoamentos de Alta Velocidade; Parte I". Relatório técnico ao IAE/CTA. Florianópolis, SINMEC/EMC/UFSC, jun. 1987. 39 p.

[7] MAJUMDAR, S. "Role of Underrelaxation in Momentum Interpolation for Calculation of Flow with Nonstaggered Grids". Numerical Heat Transfer, vol. 13, 1988. p. 125-32.

[8] VAN DOORMAAL,J.P. & RAITHBY,G.D. "Enhancements of the Simple Method for Predicting Incompressible Fluid Flows". Numerical Heat Transfer, v.7, 1984. p. 147-63.