The Emerald Research Register for this journal is available at
www.emeraldinsight.com/researchregister

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

Influences of element size and variable smoothing on inviscid compressible flow solution

C.G. Thomas and P. Nithiarasu
Civil and Computational Engineering Centre, School of Engineering, University of Wales Swansea, Swansea, UK

Received November 2003
Revised May 2004
Accepted June 2004

Abstract

Purpose – To improve inviscid compressible flow solution.

Design/methodology/approach – A local element-size calculation procedure in the streamline direction and a local variable smoothing approach are employed to improve inviscid compressible flow solution. The characteristic based split approach is used as basic solution procedure to demonstrate the employed improvements.

Findings – Results show that employing the element size in the streamline direction improves the solution accuracy in the transonic flow region. The nodal variable smoothing is very effective below a Mach number of 0.85 and produces results without any spatial oscillations.

Originality/value – This paper fills the gap by suggesting novel procedures to study Mach number range between zero and supersonic flow.

Keywords Compressible flow, Numerical analysis, Galerkin method

Paper type Research paper

1. Introduction

The characteristic based split (CBS) algorithm is now an established numerical tool for the computation of a wide range of flow problems of compressible and incompressible nature (Codina *et al.*, 1998; Nithiarasu *et al.*, 1998; Zienkiewicz *et al.*, 1999; Nithiarasu, 2003; Zienkiewicz and Codina, 1995; Zienkiewicz *et al.*, 1995; Zienkiewicz and Taylor, 2000; Zienkiewicz and Nithiarasu, 2000). The CBS scheme introduces consistent convection stabilisation, which is similar to the other available schemes such as SUPG and GLS (Lebeau *et al.*, 1993; Catabriga and Coutinho, 2002; Codina and Zienkiewicz, 2002). In the CBS scheme though, the convection stabilisation terms are controlled by the time step, which in turn is based on the stability criteria involving the local element sizes.

It is not clear, however, whether the local element size calculation methods have any significant influence on the solution. The standard element sizes employed in the past were calculated as part of the pre-processing stage and stored for use during the time stepping operation. Once calculated, these element sizes were not altered during the time stepping process. This method of evaluating the element sizes is computationally straight forward and inexpensive.

In this note, we consider the effect of using a flow dependent local element size in the streamline direction for the calculation of local time steps. Here, an updating of element sizes is required at each time step during the transient stages of the calculation. Computing the element size in the streamline direction is computationally more expensive than the standard method, especially for large-scale problems. However, the

International Journal of Numerical Methods for Heat and Fluid Flow
Vol. 15 No. 5, 2005
pp. 420-428

0961-5539
DOI 10.1108/09615530510593611

advantages gained using a stream lined element size calculation should not be overlooked. In this note, therefore, we investigate the effect of such an element size by solving inviscid compressible flow past a NACA0012 aerofoil.

In addition to the effect of element size calculation, we also address the issue of simulating inviscid flows at low Mach numbers by employing a variable smoothing approach. The employed variable smoothing approach permits oscillation free solutions at a Mach number as small as 0.01. With a combination of the element sizes in the streamline direction and the variable smoothing, inviscid solutions are obtained for Mach numbers ranging from 0.01 to 3.0.

2. The CBS scheme

The basic CBS scheme uses the characteristic Galerkin approach, first introduced for compressible flows by Zienkiewicz and Codina (1995), in which the governing flow equations are discretised in time along the characteristic. The CBS scheme consists of four steps. A fractional three-step procedure for the solution of the momentum and continuity equations followed by a fourth step, which calculates the energy distribution. The coupling between the energy and other equations are established via an equation of state. The four steps of the CBS scheme are summarised as:

(1) Solve the momentum equation without pressure terms to obtain intermediate momentum

$$\begin{aligned}\Delta U_i^* &= U_i^* - U_i^n \\ &= -\Delta t\left[\frac{\partial}{\partial x_j}(u_j U_i) - \frac{\Delta t}{2} u_k \frac{\partial}{\partial x_k}\left(\frac{\partial}{\partial x_j}(u_j U_i)\right)\right]^n + \frac{\Delta t^2}{2} u_k^n \frac{\partial}{\partial x_k}\frac{\partial p^n}{\partial x_i} \qquad (1)\end{aligned}$$

(2) Solve for pressure/density using a modified continuity equation

$$\Delta\rho = \frac{1}{c^2}\Delta p = -\Delta t\left[\frac{\partial U_i^n}{\partial x_i} + \theta_1 \frac{\partial U_i^*}{\partial x_i} - \Delta t\,\theta_1\left(\frac{\partial^2 p^n}{\partial x_i^2} + \theta_2 \frac{\partial^2 \Delta p}{\partial x_i^2}\right)\right] \qquad (2)$$

(3) Correct intermediate velocities using pressure/density calculated at step 2

$$\Delta U_i = \Delta U_i^* - \Delta t \frac{\partial p^{n+\theta_2}}{\partial x_i} \qquad (3)$$

(4) Solve the energy equation

$$\Delta(\rho E) = -\Delta t \frac{\partial}{\partial x_j} u_j(\rho E + p)^n + \frac{\Delta t^2}{2} u_k^n \frac{\partial}{\partial x_k}\left[\frac{\partial}{\partial x_j} u_j(\rho E + p)^n\right] \qquad (4)$$

The standard Galerkin approximation of the above four steps can now follow and for full details of the scheme, refer to the appropriate publications.

3. Local time stepping, variable smoothing and shock capturing

The difference between the standard CBS scheme reported previously and the one proposed here is in the calculation of the local time step. In the previous publications the local time-step at each node i, was calculated in terms of fluid and sonic velocities as:

$$\Delta t_{\text{convection}} = \frac{h_i}{|\mathbf{u}| + c} \tag{5}$$

Here, h_i is the minimum element size value for a node i. In previous papers h_i was calculated (in two-dimensions) from all the surrounding connecting elements, i.e., as (Figure 1):

$$h_i = \min(2\,\text{area}/\text{opposite side length})_{ie} \tag{6}$$

However, it is possible to calculate h_i in the streamline direction as (Figure 2)

$$h_i = \min\left(\frac{2}{\sum_{j=1}^{3} |\mathbf{s}_j \nabla \mathbf{N}_j|}\right)_{ie} \tag{7}$$

where $\mathbf{s}_j$ are the unit vectors in the streamline direction and $\mathbf{N}_j$ are the shape functions of an element ie. Note that h_i is the minimum amongst the elements connected to the node i. The time step, calculated from equation (5) is multiplied by a safety factor below 1.0 depending on the problem and mesh used.

In order to compute very low Mach number flows, a variable smoothing procedure is adopted in the place of artificial shock capturing diffusion. The following equation defines the variable smoothing applied to the transport variables, $\{\boldsymbol{\Phi}\}$, on a two-dimensional grid:

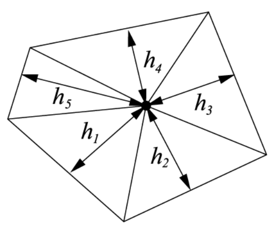


Figure 1.
Standard element size calculation

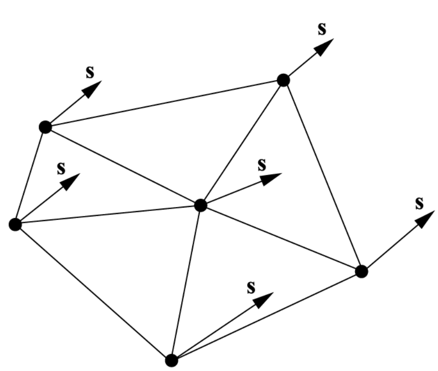


Figure 2.
Streamline element size calculation. Unit vector in the streamline direction

$$\{\boldsymbol{\Phi}\} = \left[\frac{1}{1+0.5\alpha}\{\boldsymbol{\Phi}\} + \frac{\alpha}{1+0.5\alpha}\left[\frac{\mathbf{M}-\mathbf{M}_{\mathrm{D}}}{\mathbf{M}_{\mathrm{L}}}\right]\{\boldsymbol{\Phi}\}\right] \tag{8}$$

where α is a variable smoothing parameter varies between 0 and 0.05, $\mathbf{M}$ is the consistent mass matrix, $\mathbf{M}_{\mathrm{D}}$ is the consistent mass matrix without non-diagonal terms and $\mathbf{M}_{\mathrm{L}}$ is the lumped mass matrix. By increasing α the weighting on the node in question is decreased while the influence of the surrounding nodes is increased.

At transonic and supersonic speeds, an additional shock capturing dissipation is necessary to capture shocks and to smooth local oscillations in the vicinity of shocks. A recommended shock capturing viscosity method for compressible inviscid flow problems, given by Morgan *et al.* (1990); Nithiarasu *et al.* (1998), is adopted here. For a scaler variable field ϕ the smoothed values, ϕ_{s} are computed by:

$$\left[\frac{\phi_{\mathrm{s}}^{n+1} - \phi^{n+1}}{\Delta t}\right] = \mathbf{M}_{\mathrm{L}}^{-1}\frac{C_e S_e}{\Delta t_e}(\mathbf{M} - \mathbf{M}_{\mathrm{L}})\phi^n \tag{9}$$

Here, S_e is the element "pressure switch" and is taken to be the mean of the element nodal switches S_i, which in turn are given by:

$$S_i = \frac{|\Sigma_e(p_i - p_k)|}{\Sigma_e|p_i - p_k|} \tag{10}$$

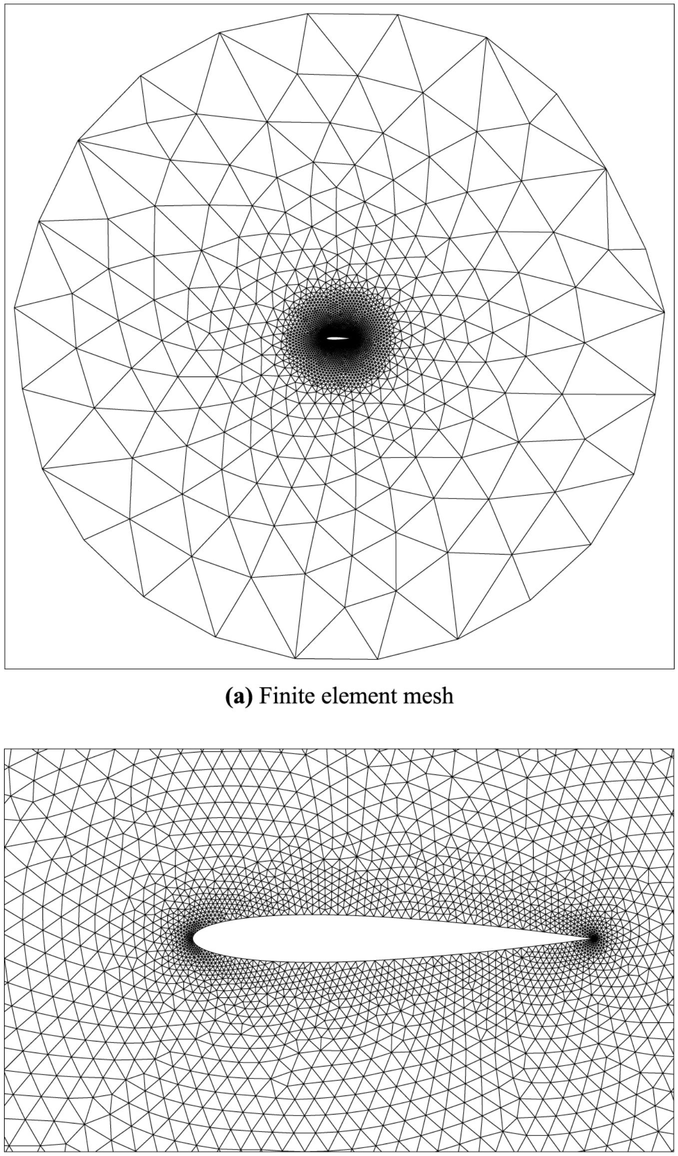
C_e is an user specified constant ranging between 0.0 and 1.0 and Δt_e is the local element time step.

4. Results

To investigate the performance of the element size calculation in the streamline direction and the local variable smoothing, an example problem of inviscid flow past a NACA0012 aerofoil is considered in this section. A Mach number range of 0.01-3.00 has been studied and results are compared with the analytical, benchmark and the standard CBS scheme solutions.

The circular computational domain used in the analysis is shown in Figure 3. The diameter of the circular domain is 25L, where L is the chord length of the NACA0012 aerofoil. Inlet conditions are prescribed on the left half of the circular boundary, and exit conditions are prescribed on the right half. An unstructured mesh consisting of 7,351 elements and 3,753 nodes is used in the domain discretisation. A close-up view of the mesh in the vicinity of the aerofoil is also shown in Figure 3. The mesh and other parameters used in the calculations are identical for the standard CBS and the modified CBS schemes.

The variation of stagnation values of density against the Mach number is shown in Figure 4. Here, it can be seen quite clearly that superior accuracy over the standard CBS scheme is achieved when an element size in the streamline direction is used, especially when the Mach number is above unity. The improvement in results at supersonic speeds may be attributed to the changes introduced by the time steps in the higher order stabilizing terms (equations (1), (2) and (4)) and shock capturing viscosity



(a) Finite element mesh

(b) Mesh details close to the aerofoil

Figure 3.
Linear triangular finite element mesh with NACA0012 aerofoil profile, 7,351 elements and 3,753 nodes

(equation (9)). However, previous experience shows that the value of shock capturing viscosity employed is almost nil at the stagnation points Nithiarasu *et al.* (1998). It is, therefore, conveniently argued that the second order terms of equations (1), (2) and (4) are responsible for the improved stagnation values. Among these terms, the second

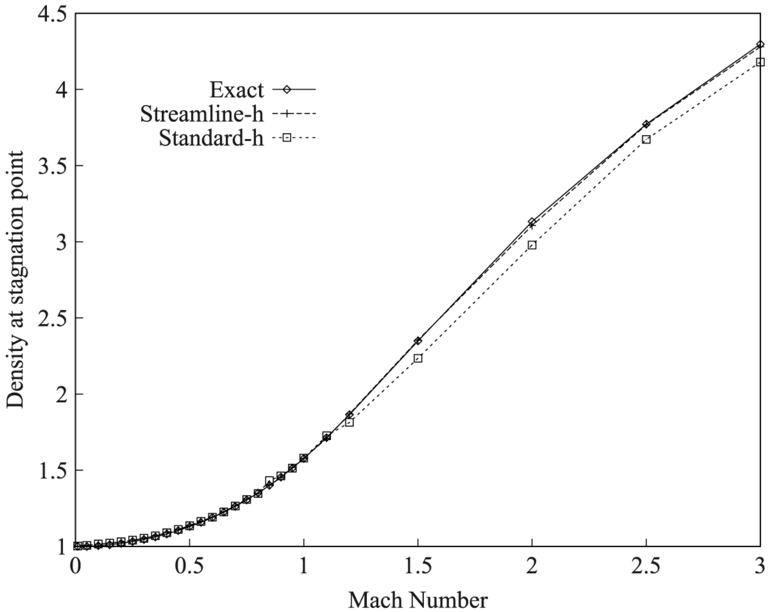


Figure 4.
Comparison of stagnation density values

order pressure term (equation (2)) multiplied by the time step directly influences the stagnation density values.

Figure 5 shows the computed density contours for a case of transonic flow at a Mach Number of 0.85 using the proposed element size calculation. The variable smoothing parameter, α, defined in equation (8) was activated when carrying out computations for Mach numbers 0.85 and below. Figure 6 shows the computed *Cp* plots over the chord length of the aerofoil. As seen, the standard CBS scheme is clearly diffusive for the same specified parameters. It is obvious, from the comparison among the benchmark structured grid data given by Pulliam and Barton (1985) and present solution that the element size calculation in the streamline direction improves the results. Here, however, the influence of the interaction between the shock capturing diffusion and element size is not ruled out, especially close to the shock. The general conclusion is that the element size in the streamline direction gives optimal shock capturing viscosity and the second order convection stabilization. It is difficult to individually quantify the effects of these terms.

To illustrate the effectiveness of the variable smoothing at low Mach numbers, we compared the solutions obtained at a Mach number of 0.25 with and without the variable smoothing and the element size calculated in the streamline direction. Without variable smoothing the non-isothermal code failed to give an accurate solution even with additional shock capturing second order diffusion. However, with variable smoothing an accurate oscillation free solution was obtained as shown in Figure 7. Comparison between the surface *Cp* distributions with and without variable smoothing are shown in Figure 8. As seen the smoothed solution is accurate and the solution without smoothing is oscillatory and wrong.

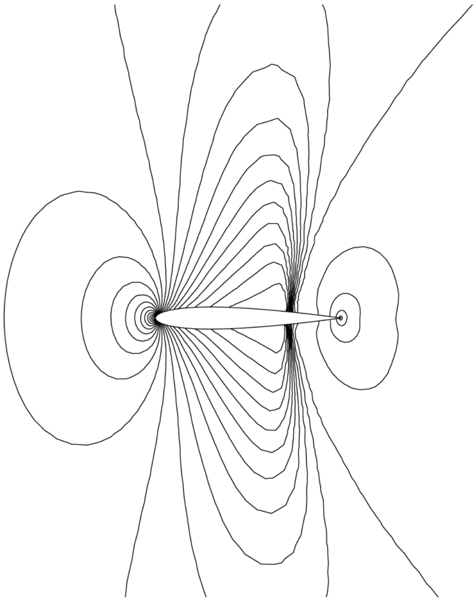


Figure 5.
Density contours for transonic flow at Mach 0.85 using proposed element size calculation

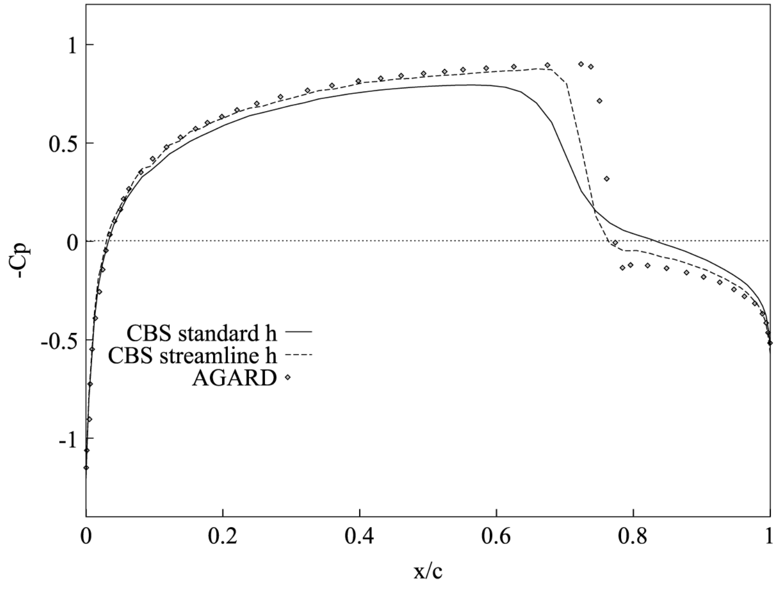


Figure 6.
Coefficient of pressure distribution at a Mach number of 0.85

5. Conclusions

A comparative study has been carried out to evaluate the performance of two local element size calculations, used in the CBS algorithm. It was found that the proposed modification in the local element size calculation, based on the streamline direction, improved accuracy, especially in the transonic region.

A flow variable smoothing algorithm was also introduced and shown to aid both schemes to enable efficient subsonic flow simulations. With the flow variable smoothing activated for Mach numbers of 0.85 or less the scheme was able to produce results without oscillations.

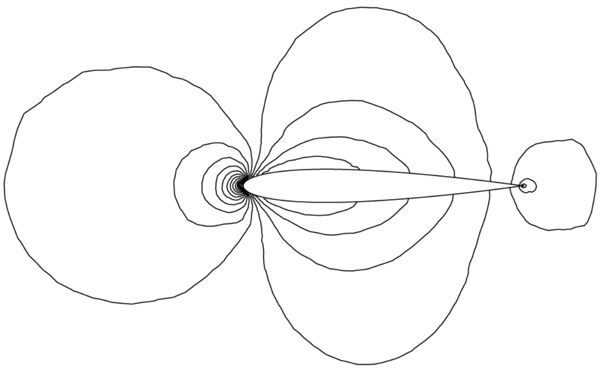


Figure 7. Density contours for transonic flow at a Mach number of 0.25 using proposed element size calculation with the addition of flow variable smoothing

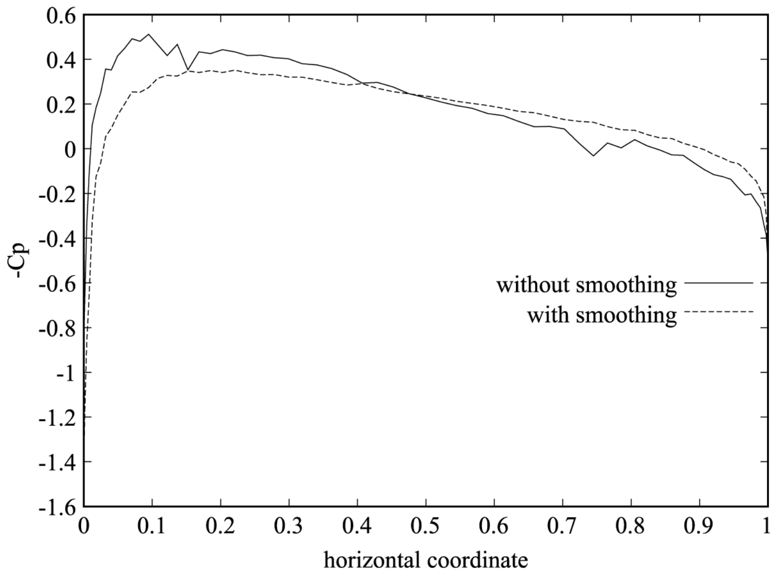


Figure 8. Coefficient of pressure plot at a Mach number of 0.25

References

Catabriga, L. and Coutinho, A.L.G.A. (2002), "Implicit SUPG solution of Euler equations using edge-based data structures", *Computer Methods in Applied Mechanics and Engineering*, Vol. 191, pp. 3477-90.

Codina, R. and Zienkiewicz, O.C. (2002), "CBS versus GLS stabilisation of the incompressible Navier-Stokes equations and the role of the time step as stabilisation parameter", *Comm. Num. Meth. Engg.*, Vol. 18, pp. 99-112.

Codina, R., Vazquez, M. and Zienkiewicz, O.C. (1998), "General algoritm for compressible and incompressible flows. Part III: a semi-implicit form", *Int. J. Num. Meth. Fluids*, Vol. 27, pp. 13-32.

Lebeau, G.J., Ray, S.E., Aliabadi, S.K. and Tezduyar, T.E. (1993), "SUPG finite element computation of compressible flows with the entropy and conservation variable formulations", *Computer Methods in Applied Mechanics and Engineering*, Vol. 104, pp. 397-422.

Morgan, K., Peraire, J., Peiro, J. and Zienkiewicz, O.C. (1990), "Adaptive remeshing applied to the solution of a shock interaction problem on a cylindrical leading edge", in Stow, P. (Ed.), *Computational Methods in Aeronautical Fluid Dynamics*, Clarenden Press, Oxford, pp. 327-44.

Nithiarasu, P. (2003), "An efficient artificial compressibility (AC) scheme based on the characteristic based split (CBS) method for incompressible flows", *Int. J. Num. Meth. Engg.*, Vol. 56, pp. 1815-45.

Nithiarasu, P., Zienkiewicz, O.C., Sai, B.V.K.S., Morgan, K., Codina, R. and Vazquez, M. (1998), "Shock capturing viscosities for the general fluid mechanics algorithm", *Int. J. Num. Meth. Fluids*, Vol. 28, pp. 1325-53.

Pulliam, T.H. and Barton, J.T. (1985), "Euler computations of AGARD working group 07 airfoil test cases", AIAA-85-0018 AIAA 23rd Aerospace Sciences Meeting, 14-17 January 1985, Reno, Nevada.

Zienkiewicz, O.C. and Codina, R. (1995), "A general algorithm for compressible flow. Part I: the split characteristic-based scheme", *Int. J. Num. Meth. Fluids*, Vol. 20, pp. 869-85.

Zienkiewicz, O.C. and Nithiarasu, P. (2000), "The characteristic-based-split (CBS) algorithm, stability and boundary conditions", *Archives of Mechanics*, Vol. 52 Nos 4-5, pp. 857-87.

Zienkiewicz, O.C. and Taylor, R.L. (2000), *The Finite Element Method*, 5th ed., Fluid Dynamics 3, Butterworth Heinemann, London.

Zienkiewicz, O.C., Morgan, K., Sai, B.V.K.S., Codina, R. and Vazquez, M. (1995), "A general algorithm for compressible flow. Part II: tests on the explicit form", *Int. J. Num. Meth. Fluids*, Vol. 20, pp. 887-913.

Zienkiewicz, O.C., Nithiarasu, P., Codina, R., Vazquez, M. and Ortiz, P. (1999), "An efficient and accurate algorithm for fluid mechanics problems. The characteristic based split (CBS) algorithm", *Int. J. Num. Meth. Fluids*, Vol. 31, pp. 359-92.

Further reading

Malan, A.G., Lewis, R.W. and Nithiarasu, P. (2002), "An improved unsteady, unstructured, artificial compressibility, finite volume scheme for viscous incompressible flows. Part I: theory and implementation", *Int. J. Num. Meth. Engineering*, Vol. 54, pp. 695-714.