INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, VOL. 17, 477–499 (1993)

ON FINITE ELEMENT APPROXIMATION AND STABILIZATION METHODS FOR COMPRESSIBLE VISCOUS FLOWS

M. FORTIN AND H. MANOUZI

Département de Mathématiques et Statistique, Université Laval, Ste-Foy, G1K 7P4, Canada

AND

A. SOULAIMANI

École de Technologie Supérieure, 4750 Av Henri Julien, Montréal, H2T 2C8, Canada

SUMMARY

This work is devoted to the numerical solution of the Navier–Stokes equations for compressible viscous fluids. Finite element approximations and stabilization techniques are addressed. We present methods to implement discontinuous approximations for the pressure and the density. An upwinding methodology is being investigated which combines the ideas behind the stream line Petrov–Galerkin method and the flux limiter methods aiming to introduce numerical diffusion only where it is necessary.

1. INTRODUCTION

This paper presents some results for the numerical solution of the Navier–Stokes equations for compressible fluids. We shall indeed address a few basic issues such as the choice of elements or the use of upwinding or some other form of artificial viscosity. Some of the methods presented are relatively non-standard and may not be, for the moment, competitive with more established techniques. Nevertheless, we believe that in the present state of the art, it is worth exploring alternative routes that might ultimately lead to more insight.

Our main concern will be the treatment of the density transport equation in which no natural dissipative term is present and for which standard finite element techniques seem ill-adapted. Experience has shown that numerical results, specially for moderately large Mach numbers, are often spoiled by spurious oscillations of pressure, density and temperature, these three variables being linked by the state equation. Those oscillations strikingly look like the famous 'checker-board patterns' of the incompressible case and one wonders whether they have the same origin. We have already discussed in another paper[1] the importance of the choice of stable elements. The essential conclusion is that either one should employ an element which is suitable for the incompressible case or some form of stabilization technique derived from this case. Examples of suitable elements and discussion of stabilization methods can be found in References 2–5. However, in presence of shocks or strong gradients, even stable elements yield oscillatory results. At present, the only cure seems to be the introduction of some form of artificial viscosity. This can

0271–2091/93/180477–23$16.50

Received April 1991
Revised November 1992

be done in various ways, explicit or implicit. Upwinding can, in this respect, be considered as a way of introducing artificial viscosity.

We consider various adaptations of the streamline upwinding Petrov–Galerkin method.[6–8] Our originality here will be to incorporate some ideas related to the flux limiters or slope limiters techniques currently employed in the finite difference method for the approximation of the Euler equations.

2. FORMULATION OF THE PROBLEM

2.1. General presentation

Let Ω be a bounded domain of $\mathbb{R}^2$ or $\mathbb{R}^3$ and let $\Gamma=\partial\Omega$ be its boundary. We use the following non-conservative form of the Navier–Stokes equations:

$$\frac{\partial \rho}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,\rho+\rho\,\mathrm{div}\,\mathbf{u}=0, \tag{1}$$

$$\rho\left[\frac{\partial \mathbf{u}}{\partial t}+(\mathbf{grad}\,\mathbf{u})\mathbf{u}\right]-\mathrm{div}\,\tau+\mathbf{grad}\,p=\mathbf{f}, \tag{2}$$

$$\rho\left(\frac{\partial T}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,T\right)-\frac{\gamma}{Re\,Pr}\Delta T-p\,\mathrm{div}\,\mathbf{u}=\mathbf{\Phi}, \tag{3}$$

$$\tau=\frac{1}{Re}\left[\mathbf{grad}\,\mathbf{u}+(\mathbf{grad}\,\mathbf{u})^{\mathrm{T}}-\frac{2}{3}(\mathrm{div}\,\mathbf{u})\mathbf{I}\right], \tag{4}$$

where p,ρ and T are linked by the state equation

$$F(p,\rho,T)=0. \tag{5}$$

In (3), Φ represents the viscous heat production. These equations must of course be completed by boundary conditions. Figure 1 presents a typical set of such conditions corresponding to the problem for which we present numerical results. On the outflow part of the boundary, we used a no-stress condition which is of course artificial. We also consider steady-state problems but in our computations this will be done by making a time-dependent solution converge.

Our starting point in studying the compressible viscous Navier–Stokes equations will be the use of the variable $\sigma=\log\rho$ instead of density. Dividing equation (1) by ρ, which is assumed positive non-zero, and using this change of variable, we obtain indeed a suitable form of the continuity equation, containing a convective term in σ:

$$\mathrm{div}\,\mathbf{u}+\mathbf{u}\cdot\mathbf{grad}\,\sigma+\frac{\partial\sigma}{\partial t}=0. \tag{6}$$

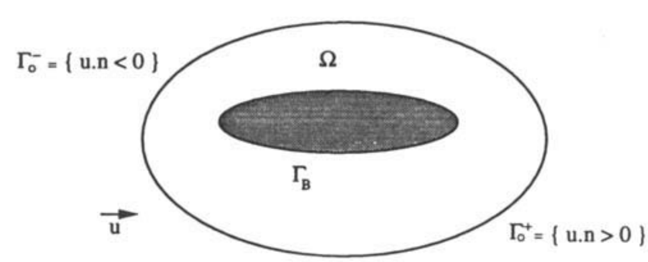

Figure 1. Domain Ω and boundary conditions

It is clearly seen from (6) that there is no diffusion process that might control the transport of σ, specially near strong variations zones such as shocks. The Navier–Stokes equations for $\mathbf{u}, \rho, \sigma$ and T are then written as follows:

$$
\begin{gathered}
\mathrm{e}^{\sigma}\left[\frac{\partial \mathbf{u}}{\partial t}+(\mathbf{grad}\,\mathbf{u})\mathbf{u}\right]-\operatorname{div}\boldsymbol{\tau}+\mathbf{grad}\,p=\mathbf{f}, \\
\frac{\partial \sigma}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,\sigma+\operatorname{div}\mathbf{u}=0, \\
\mathrm{e}^{\sigma}\left[\frac{\partial T}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,T\right]-\frac{\gamma}{Re\,Pr}\Delta T-p\operatorname{div}\mathbf{u}-\Phi=0,
\end{gathered}
\tag{7}
$$

while the state equation now relates σ with pressure and temperature

$$\sigma=f(p, T). \tag{8}$$

For example, in the case of a perfect gas, (8) becomes

$$\sigma=\log\frac{p}{(\gamma-1)T},$$

and for an isentropic flow

$$\sigma=\frac{1}{\gamma}\log p/c_0,$$

where c_0 is a constant.

As we are interested in looking for finite element solutions of system (7), we consider the following variational problem. Find $\mathbf{u}, \sigma, p$ and T belonging to suitable spaces and satisfying

$$
\begin{gathered}
\left(\mathbf{v}, \mathrm{e}^{\sigma}\frac{\partial \mathbf{u}}{\partial t}\right)+c(\mathbf{u},\mathbf{u},\mathbf{v}\mathrm{e}^{\sigma})+b(\mathbf{v},p)+\frac{1}{Re}a(\mathbf{u},\mathbf{v})=(\mathbf{f},\mathbf{v})-b(\mathbf{u},q)+\left(q,\frac{\partial \sigma}{\partial t}\right)+d(q,\sigma)=0, \\
\left(\psi\mathrm{e}^{\sigma},\frac{\partial T}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,T\right)+\frac{\gamma}{R_e\,P_r}(\mathbf{grad}\,\psi,\mathbf{grad}\,T)-(\psi,p\operatorname{div}\mathbf{u}+\Phi)=0,
\end{gathered}
\tag{9}
$$

for all test functions $\mathbf{v}, q$ and ψ. Here the notations $(\cdot,\cdot)$ stands for the $L^2(\Omega)$ scalar product. Given a partition $\Omega=\bigcup K$, we use $(\cdot,\cdot)_K$ for the $L^2(K)$ scalar product. The multilinear forms $a(\cdot,\cdot)$, $b(\cdot,\cdot)$ and $c(\cdot,\cdot,\cdot)$ in (9) are defined by

$$
\begin{gathered}
c(\mathbf{u},\mathbf{u},\mathbf{v})=[\mathbf{v},(\mathbf{grad}\,\mathbf{u})\mathbf{u}] \\
b(\mathbf{v},p)=-(p,\operatorname{div}\mathbf{v}) \\
a(\mathbf{u},\mathbf{v})=\int_\Omega \mathbf{grad}\,\mathbf{v}:\{\mathbf{grad}\,\mathbf{u}+[(\mathbf{grad}\,\mathbf{u})^T-\tfrac{2}{3}(\operatorname{div}\mathbf{u})\mathbf{I}]\}\,\mathrm{d}x.
\end{gathered}
\tag{10}
$$

The bilinear form $d(q,\sigma)$ will be written in various ways depending on the discretization of $\mathbf{u}\cdot\mathbf{grad}\,\sigma$, in fact on the approximation of σ and eventually on the upwinding method used. We present an implementation of discontinuous pressure and density approximations which is, to our knowledge, a new feature in the context of compressible flows. The expected counterpart is a better reproduction of strong gradients and a gain of computational efficiency. On the other hand, continuous approximations are very often used in practice and straightforward for implementation. In both cases, oscillations appear for relatively low Reynolds and Mach numbers, whenever the expression of $d(\sigma, q)$ leads to a centred approximation of $\mathbf{u}\cdot\mathbf{grad}\,\sigma$. For instance, when using a Galerkin formulation, $d(\sigma, q)$ is simply the scalar product $(q, \mathbf{u}\cdot\mathbf{grad}\,\sigma)$ which is known to generate a centred approximation. To avoid such an undesirable difficulty we shall use modified expressions of $d(q, \sigma)$ containing, in one way or other, numerical diffusion.

This being said, one is naturally led to choose for the approximation of σ the same type of elements as for p. However, this is not essential if one introduces a decoupled formulations such as (6) and (7) and making a different choice would be equivalent to a weak treatment of the state equation.

Looking at equation (6), one sees a transport equation without any diffusive term. We must therefore discretize it in a correct way, avoiding in particular the usual pitfalls associated with centred schemes.

Among the many possible choices of suitable discretizations, we consider only triangular meshes where velocity is approximated by piecewise continuous quadratic (P_2) elements and temperature by piecewise linear (P_1) continuous elements. Pressure and density are represented either by a continuous or by a discontinuous piecewise linear approximation. In the case of discontinuous linear pressure, a bubble must be added to the velocity field to yield a stable approximation of incompressible problems.

However, a discontinuous field for σ leads to difficulties with the term $d(q, \sigma) = (\mathbf{u}\cdot\mathbf{grad}\,\sigma, q)$ which is now undefined. We must therefore look for some way of circumventing the problem. On the other hand, standard continuous approximations naturally lead to a centred approximation of (6) or related equations. This is well known to be ill-suited for a transport equation. We present in the next section some ways to circumvent those difficulties.

3. DISCRETIZATION OF THE CONVECTIVE TERM $\mathbf{u}\cdot\mathbf{grad}\,\sigma$

This section is devoted to the presentation of different options for the discretization of the convective term in the transport equation for σ. The possibilities are different for continuous or for discontinuous approximations. The classical method for the stabilization of scheme for the transport equation relies on upwinded differences; for one-dimensional problems this is fairly well established and known to be equivalent to introducing an artificial viscosity. We try to follow this line and introduce some upwinding methods. The first is based on a simple numerical directional derivative. In the second method we try to combine the streamline upwinding method with a flux limiter in order to restrict artificial viscosity to zones where it is really needed. Finally, we present methods adapted to handle discontinuous pressure approximations.

3.1. Continuous approximations

3.1.1. Directional derivative method. This method is based on the very simple idea that the convective term in a transport equation can be interpreted as a directional derivative. This directional derivative will in turn be computed for each node by an upwind finite difference scheme along the streamline direction $\mathbf{u}$. Let us introduce some notation (see Figure 2).

Let σ_i be the value of σ at node i and $\|\mathbf{u}_i\|$ be the Euclidian norm of the velocity at this same node. We denote P_i^- (resp. P_i^+) the upwind (resp. downwind) streamline projection of node i on the boundary ∂K, where the element K is one of the elements sharing node i. Then h_i^-(resp. h_i^+) is the distance between node i and point P_i^- (resp. P_i^+), while σ_i^- (resp. σ_i^+) is the value of σ at point P_i^- (resp. P_i^+).

Using the above definitions we define the upwind flux at node i by

$$(\mathbf{u}\cdot\mathbf{grad}\,\sigma)_i^- = \|\mathbf{u}_i\|\frac{\sigma_i - \sigma_i^-}{h_i^-}, \tag{11}$$

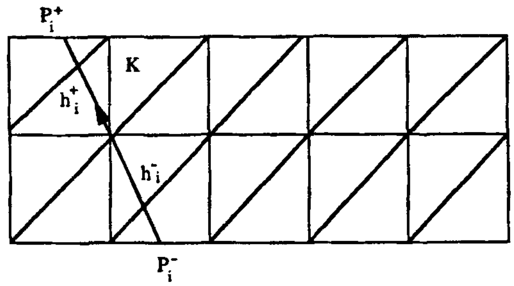

Figure 2.

while, in a similar way, the downwind flux is given by

$$(\mathbf{u}\cdot\mathbf{grad}\,\sigma)_i^+ = \|\mathbf{u}_i\|\frac{\sigma_i^+ - \sigma_i}{h_i^+}\ . \tag{12}$$

Inside element K a discretization of the flux is then expressed using a simple combination of the nodal fluxes:

$$\begin{aligned}\int_K (\mathbf{u}\cdot\mathbf{grad}\,\sigma)q\,\mathrm{d}x &= \sum_i \int_K q_i\,[\alpha(\mathbf{u}\cdot\mathbf{grad}\,\sigma)_i^- + (1-\alpha)(\mathbf{u}\cdot\mathbf{grad}\,\sigma)_i^+]\,\mathrm{d}x\ ,\\ &= [q, (\mathbf{u}\cdot\mathbf{grad}\,\tilde{\sigma})]_K\ ,\end{aligned} \tag{13}$$

where the sum is taken on the vertices of K, q is the standard linear interpolation function and the parameter α is chosen to adjust the artificial diffusion. We then define the bilinear form $d(\cdot,\cdot)$ in (9) by

$$d(q,\sigma)_{\mathrm{DD}} = \sum_K [q, (\mathbf{u}\cdot\mathbf{grad}\,\tilde{\sigma})]_K\ . \tag{14}$$

However, this method is only first-order accurate and relatively diffusive as will be shown in the numerical results. It would be possible to build higher-order versions of this method by using more points in the upwind direction. This idea has been also studied by Tabata and Yaoi.[9]

3.1.2. Streamline upwinding flux limiter method (SUFL). Streamline–diffusion methods have been very popular in recent years for the solution of transport equations.[5,8] We consider it here only in its simplest implementation. However, we try to incorporate in the method ideas coming from flux limiters methods. The essential idea is to introduce streamline diffusion only where it is necessary. Since upwinding methods modify the original problem by adding diffusion, accuracy is affected. Flux limiter methods have been extensively developed in the context of finite differences and finite volume methods.[10] Their principle consists in computing a sensor parameter which indicates strong gradients zones where an amount of numerical diffusion is added to the higher-order centred scheme. In a standard (consistent) streamline–diffusion method, one would define the bilinear form $d(q,\sigma)$ by

$$d(q,\sigma)_{\mathrm{SU}} = \int_\Omega q\,\mathbf{u}\cdot\mathbf{grad}\,\sigma\,\mathrm{d}x + \int_\Omega \beta(\mathbf{u}\cdot\mathbf{grad}\,q)\left(\mathrm{div}\,\mathbf{u} + \mathbf{u}\cdot\mathbf{grad}\,\sigma + \frac{\partial\sigma}{\partial t}\right)\mathrm{d}x\ . \tag{15}$$

This formulation is consistent in the sense that the extra term vanishes for the exact solution of (9). In (15), one usually takes $\beta = h/2\|\mathbf{u}\|$. We define

$$\beta = \frac{h}{2\|\mathbf{u}\|}\zeta(r)\ ,$$

where r is a sensor parameter defined for each node i, using the notation of the previous section, by

$$r_i = \frac{(\mathbf{u} \cdot \mathbf{grad}\, \sigma)_i^-}{(\mathbf{u} \cdot \mathbf{grad}\, \sigma)_i^+} \,. \tag{16}$$

Intuitively one may get negative values for the sensor parameter near shocks or oscillations, indicating zones where maximum diffusion has to be added. On the contrary $r \approx 1$ indicates a smooth region. Therefore, the function $\zeta(r)$ should take its maximum value for $r \leq 0$ and zero for $r \geq 1$. For $0 \leq r \leq 1$, many choices are *a priori* possible. For instance we may take $\zeta(r) = 1 - r$. For one-dimensional finite difference scheme, it can be shown that $\zeta(r)$ must satisfy some conditions in order to keep high-order convergence.[10] We tested the following definition of the asymptotic like function $\zeta(r)$

$$\zeta(r) = \begin{cases} 1, & r \leq 0, \\ 1 - 2r/(1+r), & 0 \leq r \leq 1, \\ 0, & r \geq 1. \end{cases} \tag{17}$$

Remark 1

Integrating by parts $d(q, \sigma)$ as defined by (15), it can easily be shown that the problem corresponding to the modified continuity equation can be written as

$$\operatorname{div} \mathbf{u} + \frac{\partial \sigma}{\partial t} + \mathbf{u} \cdot \mathbf{grad}\, \sigma - \operatorname{div} [\beta \mathbf{u}(\operatorname{div} \mathbf{u} + \frac{\partial \sigma}{\partial t} + \mathbf{u} \cdot \mathbf{grad}\, \sigma)] = 0 \,.$$

The streamline upwinding flux limiter flux is a residual method such as the SUPG method. Thus, it maintains the high-order accuracy of Galerkin's method (for smooth solutions) while it improves stability. Furthermore, the SUFL method is intrinsically non-linear, i.e. the stabilization process acts like in any 'shock-capturing' method. Zones of strongly varying fluxes, where oscillations are most likely to appear, are first detected and the stabilization takes place by smoothing the solution.

Remark 2

Many choices of the sensor parameter or the diffusion function ζ are possible and only numerical experiment can discriminate their relative performance. This choice should at least respect the following criteria: smooth variations of the sensor parameter and the doubly asymptotic behaviour of the diffusion function. Particularly, in the case of small nodal upwind and downwind gradients the value of r should be close to one in order to avoid any computational oscillation.

3.2. Discontinuous discretization

We now consider the use of a discontinuous approximation for σ and p. In our computations, this approximation was piecewise linear but the methods described are general and could be applied to a wide class of other approximations.

3.2.1. Smoothing techniques. One obvious method which may be thought of, consists in finding a continuous field $\tilde{\sigma}$ by smoothing σ, that is by taking $\tilde{\sigma}$ to be the solution of the problem

$$(q, \tilde{\sigma} - \sigma) = 0 \qquad \forall q \in Q \,, \tag{18}$$

where Q is the space of piecewise continuous linear elements. This is nothing but a $L^2(\Omega)$ projection of the discontinuous field σ onto Q. Afterwards, $d(q,\sigma)$ could simply be defined by

$$d(q,\sigma)=\int_\Omega q(\mathbf{u}\cdot\mathbf{grad}\,\tilde{\sigma})\mathrm{d}x\,. \tag{19}$$

We rather used a variant of this which consists in looking for the $L^2(\Omega)$ projection of $\mathbf{u}\cdot\mathbf{grad}\,\sigma$, this last expression being taken in the sense of distributions. We thus solve the problem:

$$(q,\tilde{\phi})=\langle\mathbf{u}\cdot\mathbf{grad}\,\sigma,q\rangle\quad\forall q\in Q\,. \tag{20}$$

Since q is continuous, we can define, from (20),

$$d(q,\sigma)_{\mathrm{SM}}=\sum_K\int_K\left[q(\mathbf{u}\cdot\mathbf{grad}\,\sigma)\,\mathrm{d}x-\int_{\partial K}q\mathbf{u}\cdot\mathbf{n}\sigma\,\mathrm{d}\gamma\right]+\int_\Gamma q(\mathbf{u}\cdot\mathbf{n})\sigma_{\mathrm{ext}}\,\mathrm{d}\gamma, \tag{21}$$

where $\mathbf{n}$ is the outward normal direction to ∂K the boundary of K.

Formula (21) takes into account the jump of σ at inter-element boundaries and σ_{ext} is either a given boundary condition on the inflow part of Γ or taken equal to the interior value (to make the jump vanish) on the outflow part.

3.2.2. Lesaint–Raviart method. A second method dealing with a discontinuous density would not use any smoothing technique. It was introduced by Lesaint and Raviart[1] and it also contains an upwinding effect.

For each element K, let ∂K^- be the inflow part of the boundary ∂K of K (see Figure 3),

$$\partial K^-=\{x\in\partial K,\quad \mathbf{u}\cdot\mathbf{n}(\mathbf{x})<0\}\,.$$

We can now take q belonging to the space of discontinuous piecewise linear polynomials and we can write

$$d(q,\sigma)_{\mathrm{LR}}=\sum_K\left(\int_K(\mathbf{u}\cdot\mathbf{grad}\,\sigma)q\,\mathrm{d}x+\int_{\partial K-}q\mathbf{u}\cdot\mathbf{n}[[\sigma]]\,\mathrm{d}\gamma\right), \tag{22}$$

where $[[\sigma]]=\sigma_{\mathrm{ext}}-\sigma$ is the jump of σ across the element boundary.

This expression can be interpreted as a local approximation of a derivative in the sense of distributions while the smoothing technique of the previous section was globally defined on the whole of Ω. In this sense the smoothing technique can also be considered as an averaged Lesaint–Raviart method.

This method has been proven to yield stable and convergent scheme for the transport-diffusion equation.[5] Furthermore it has the interesting property to intrinsically introduce numerical dissipation.[11] However, its application to flow problem was deceptive. A preliminary computation of transonic flow shows existence of oscillations, which even blow up for supersonic regime.

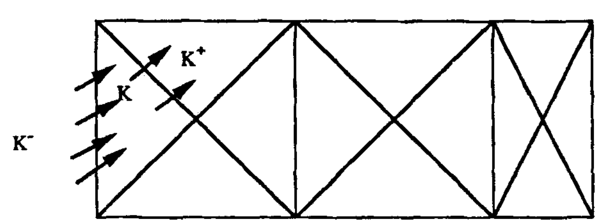

Figure 3.

We are led to examine more carefully this scheme particularly for triangular elements and for really discontinuous solutions. To this purpose let us study the following benchmark example. Consider the case of a given discontinuous field σ in a two-dimensional triangular mesh (Figure 4) and a constant one-dimensional velocity $\mathbf{u}$.

One computes $d(q, \sigma)$ using relation (22). The first integral vanishes since σ is constant for each element, hence $d(q, \sigma)$ is non-zero only for elements which share one side with the interface of discontinuity. The pattern of the values of $d(q, \sigma)$ is, as sketched in Figure 5, non-uniform in the transverse direction even if the problem is really one-dimensional.

This fact shows the strong dependency of the scheme to the mesh and its incapacity to represent discontinuous fields. Hence, in the presence of shocks the Lesaint–Raviart scheme would lead to irregularities of the right-hand side of the variational problem (9) which dramatically reflects on velocity and on the pressure. Comparing the smoothing method introduced in Section 3.1.1, one can note the following interesting remark.

The projection $\tilde{\phi}$ of $\mathbf{u}\cdot\mathbf{grad}\,\sigma$ over continuous P_1 functions is given by the following equation:

$$(q, \tilde{\phi}) = \sum_K \left(\int_K q(\mathbf{u}\cdot\mathbf{grad}\,\sigma)\,\mathrm{d}x - \int_{\partial K} (\mathbf{u}\cdot\mathbf{n})\,\sigma q\,\mathrm{d}\gamma \right).$$

Unlike in Lesaint–Raviart scheme, all elements contribute to the calculation of $\tilde{\phi}$ and thus to $d(q,\sigma)$ as indicated through relation (21). The consequence of this process is in fact more smoothness and regularity of the functional $d(q,\sigma)_{\mathrm{SM}}$. Indeed it can be easily shown that this 'averaged Lesaint–Raviart method' provides constant solution $\tilde{\phi}$, in the transverse direction, when applied to the benchmark example discussed above. The counterpart of this gain of smoothness is the loss of upwinding and more artificial dispersion.

4. GENERALIZATION OF THE SUFL METHOD

So far we discussed various upwinding methods and showed how to apply them for the continuity equation. All of the above methods define the density gradient in the variational formulation in

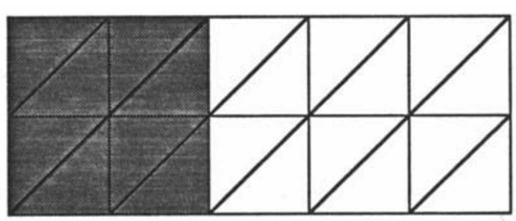

Figure 4. The test triangular mesh and its velocity field

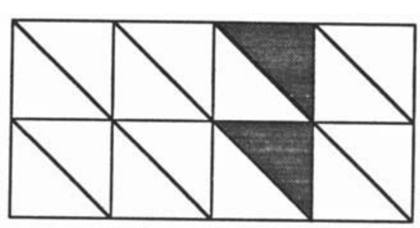

Figure 5. Values of $d(q, \sigma)$

order to damp high-frequency density oscillations. This is effective to control steep density gradients for relatively high Mach number flows but more diffusion is also needed to stabilize the velocity components and the temperature in case of high Reynolds number flows. In this section we focus particularly on the SUFL method, applying it to the whole the set of equations (1)–(3). The corresponding variational problem will read as follows:

$$\left(\mathbf{v},\frac{\partial \mathbf{u}}{\partial t}\mathrm{e}^{\sigma}\right)+c(\mathbf{u},\mathbf{u},\mathbf{v}\,\mathrm{e}^{\sigma})+b(\mathbf{v},p)+\frac{1}{Re}a(\mathbf{u},\mathbf{v})+e(\mathbf{v},\mathbf{u})=(\mathbf{f},\mathbf{v})-b(\mathbf{u},q)+\left(q,\frac{\partial \sigma}{\partial t}\right)+d(q,\sigma)=0$$

$$\left(\psi\mathrm{e}^{\sigma},\frac{\partial T}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,T\right)+\frac{\gamma}{Re\,Pr}(\mathbf{grad}\,\psi,\mathbf{grad}\,T)-(\psi,p\,\mathrm{div}\,\mathbf{u}+\Phi)+h(\psi,T)=0, \qquad (23)$$

where

$$e(\mathbf{v},\mathbf{u})=\sum_{K}\int_{K}\left\{(\mathbf{u}\cdot\mathbf{grad}\,\mathbf{v})\,\Lambda\left[\mathrm{e}^{\sigma}\left(\frac{\partial \mathbf{u}}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,\mathbf{u}\right)-\mathrm{div}\,\boldsymbol{\tau}+\mathbf{grad}\,p-\mathbf{f}\right]\right\}\mathrm{d}x. \qquad (24)$$

In the same way, we set

$$h(\psi,T)=\sum_{K}\int_{K}\left\{(\mathbf{u}\cdot\mathbf{grad}\,\psi)\,\Lambda_T\left[\mathrm{e}^{\sigma}\left(\frac{\partial T}{\partial t}+\mathbf{u}\cdot\mathbf{grad}\,T\right)-\frac{\gamma}{Re\,Pr}\Delta T-p\,\mathrm{div}\,\mathbf{u}-\Phi\right]\right\}\mathrm{d}x. \qquad (25)$$

In (24) and (25) we have

$$\Lambda_{ij}=\begin{cases}0 & \text{for } i\neq j,\\ \dfrac{h[r(u_i)]}{2\,\|\mathbf{u}\|\,\zeta} & \text{for } i=j,\end{cases}$$

$$\Lambda_T=\frac{h}{2\,\|\mathbf{u}\|}\zeta[r(T)].$$

In this formulation the second-order derivative terms are computed inside the elements. Therefore, we face the following dilemma: for linear approximations these terms completely disappear and it is a sign of the poor accuracy of this method. On the other hand, for higher-order approximations its implementation is cumbersome. In order to circumvent this problem, we propose to slightly modify the above formulation. *The divergence of the diffusion flux vector will be*

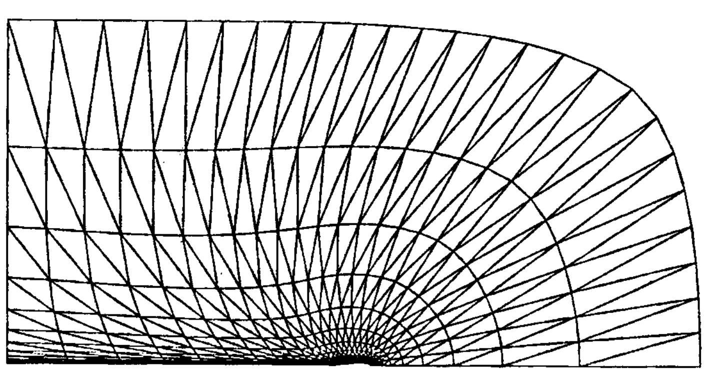

Figure 6. The mesh used in the tests

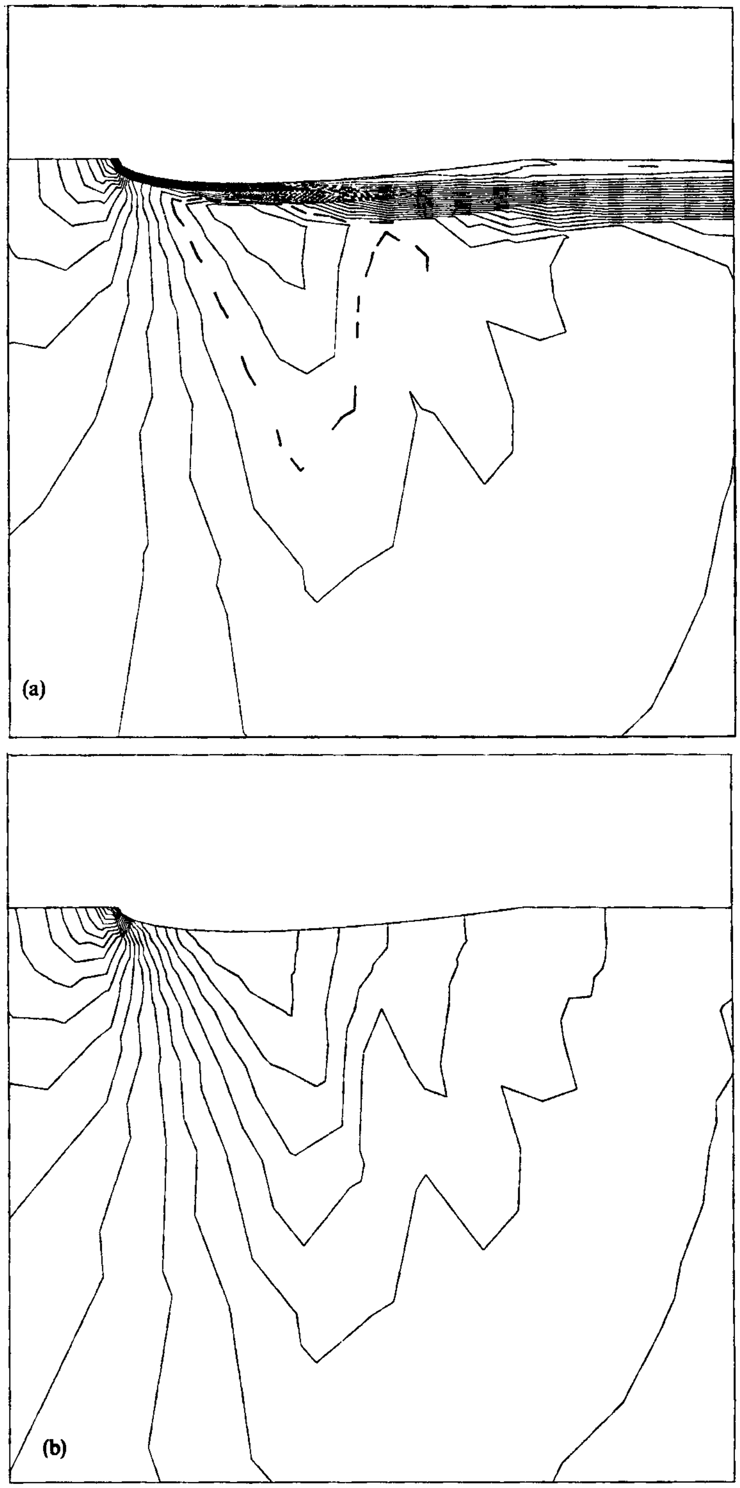

Figure 7. Continuous approximation, no upwinding: (a) isomach contours; (b) isobar contours; (c) stream vectors

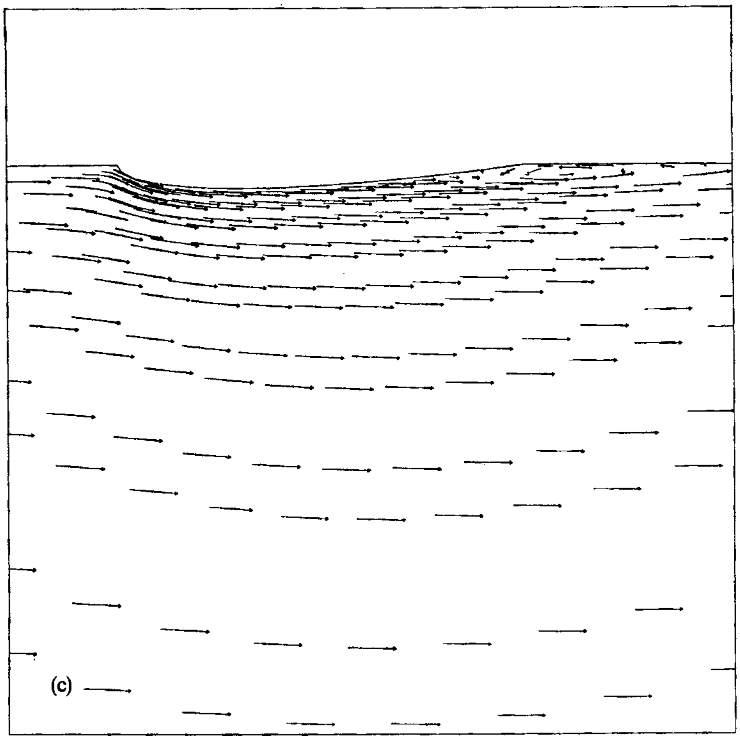

Figure 7. (Continued)

in fact approximated by its projection over continuous P_1 functions using the same techniques as in the approximation of the convective terms for discontinuous approximations.

5. SOLUTION ALGORITHM

Problem (9) involves two different types of non-linearities: convective-like non-linearities, which are present in the incompressible case as well, and compressibility-related non-linearities which appear in the state equation and through the high coupling of velocity, pressure, temperature and density. Newton's algorithm, coupled with any implicit Euler time discretization scheme, may be thought to be effective for solving such problems due to its interesting convergence properties. However, in the present problem, the implementation is not obvious and practically impossible, so that we are led to look for other useful variants. To this end, a quasi-Newton method has been developed based essentially on the generalized minimum residual method.[12] Indeed to be effectively implemented, Newton's method requires the computation of the first-order variations of the functionals $c(\cdot,\cdot,\cdot)$ and $d(\cdot,\cdot)$ for any small perturbation of the solution. However, it is difficult to get the exact analytical expressions for those functionals and for the corresponding tangent matrix when upwinding techniques are used. Hence, only approximations could really be written. On the other hand, solving system (9) directly requires large storage if one thinks of industrial applications. To circumvent those difficulties, we use a non-linear variant of the GMRES algorithm.[12–14] This algorithm does not require any matrix computation but only computation of successive residuals and scalar products. We give no more details as this is not a really original feature of our computations.

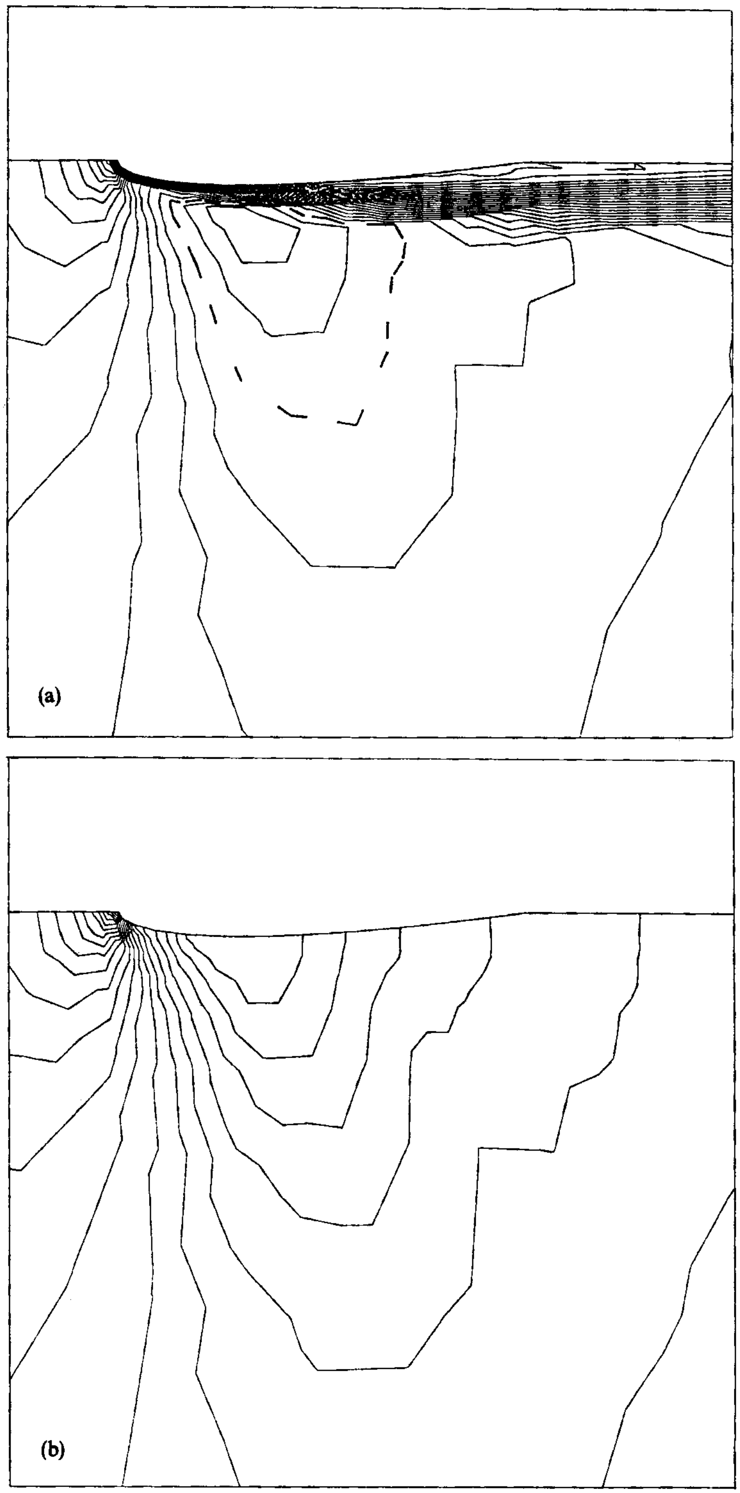

Figure 8. SUFL, isentropic case: (a) isomach contours; (b) isobar contours; (c) stream vectors

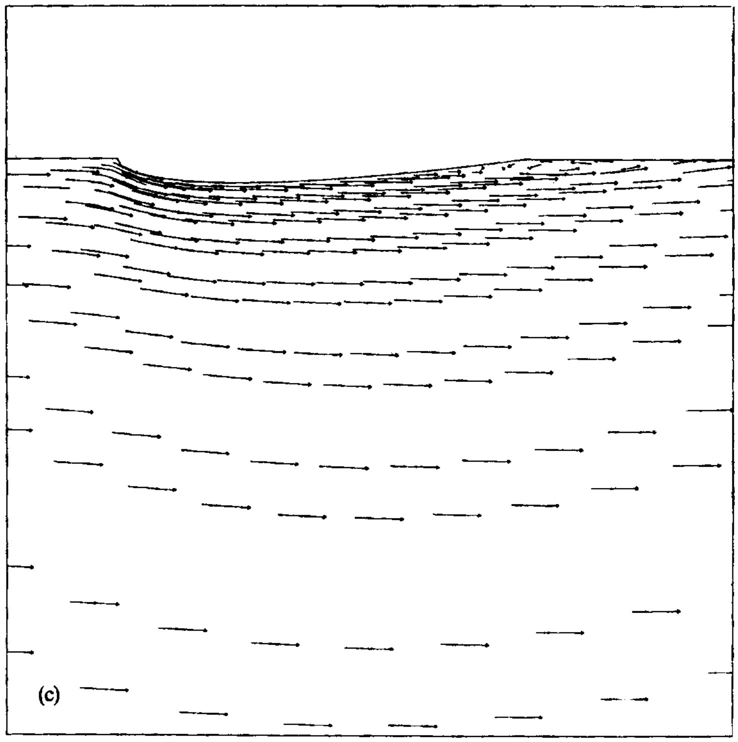

Figure 8. (Continued)

6. NUMERICAL RESULTS

A few computations have been carried for simple flow situations in order to check and compare the qualities of the methods developed in the previous sections. We present results for a laminar flow at $Re = 1500$ around a NACA0012 aerofoil with zero angle of attack and at a Mach number $M = 0{\cdot}85$. The mesh used is relatively coarse and is shown in Figure 6.

The first test is performed for an isentropic flow. Therefore, temperature has no effect on the other variables and the various upwinding methods on density could be properly and easily compared. For every test case, we present isomach and isobar contours and stream vector plots.

Figures 7(a)–7(c) present results for the case of a P2–P1 approximation without any upwinding oscillations are clearly apparent and this result is used as the comparison reference.

Figures 8(a)–8(c), 9(a)–9(c) and 10(a)–10(c) present results (with the same isentropic flow) for various upwinding methods applied to the density equation. In Figure 8 we see the result of the SUFL method of Section 3.1.2, which reduces oscillations but does not eliminate them completely. The directional derivative method of Section 3.1.1 (Figure 9) performs better in this respect but is clearly too diffusive. In particular, it completely kills the vortex at the trailing edge. Finally, the smoothing method of Section 3.2.1 stands somewhere in between, reducing oscillations but keeping the vortex at the trailing edge, although strongly reduced.

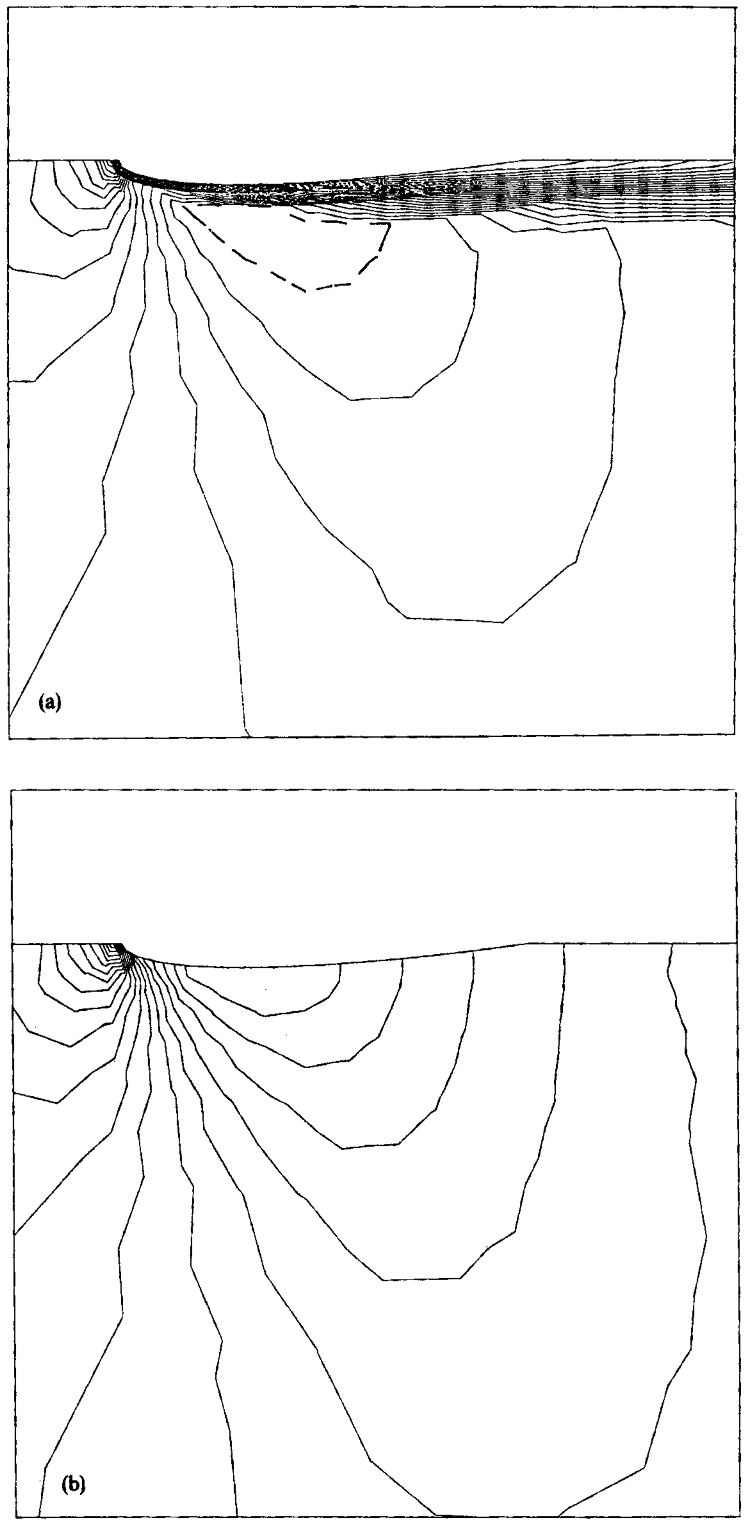

Figure 9. Directional derivative, isentropic case: (a) isomach contours; (b) isobar contours; (c) stream vectors

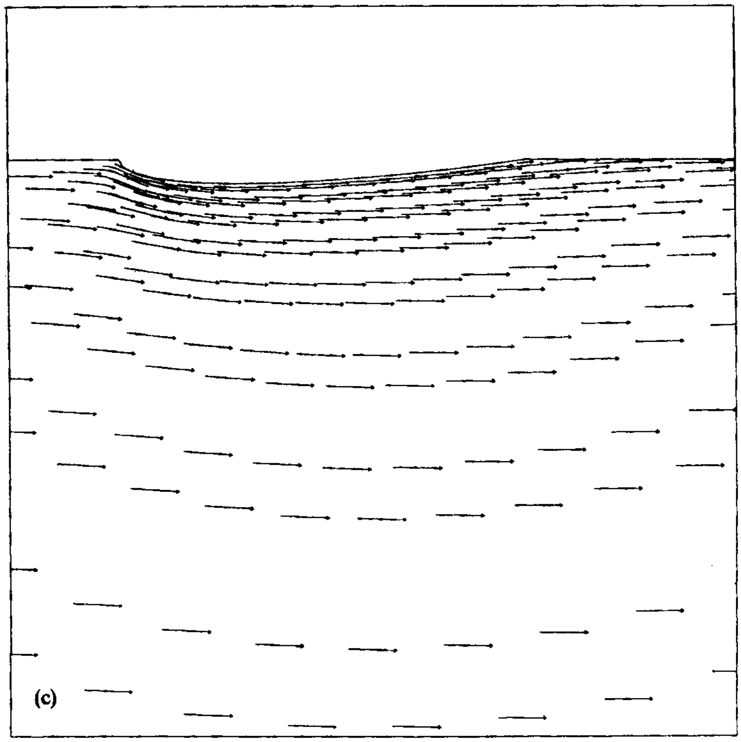

Figure 9. (Continued)

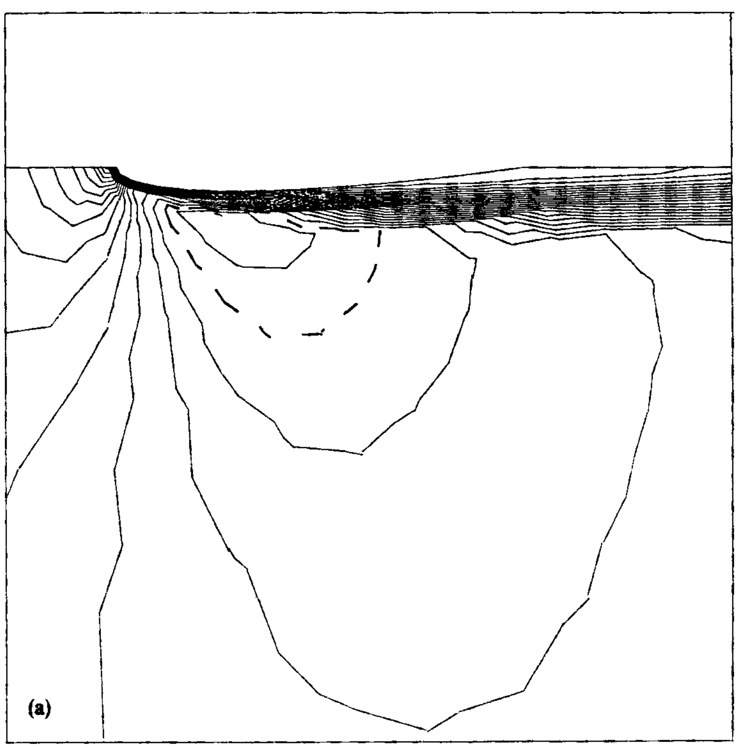

Figure 10. Discontinuous smoothing method, isentropic case: (a) isomach contours; (b) isobar contours; (c) stream vectors

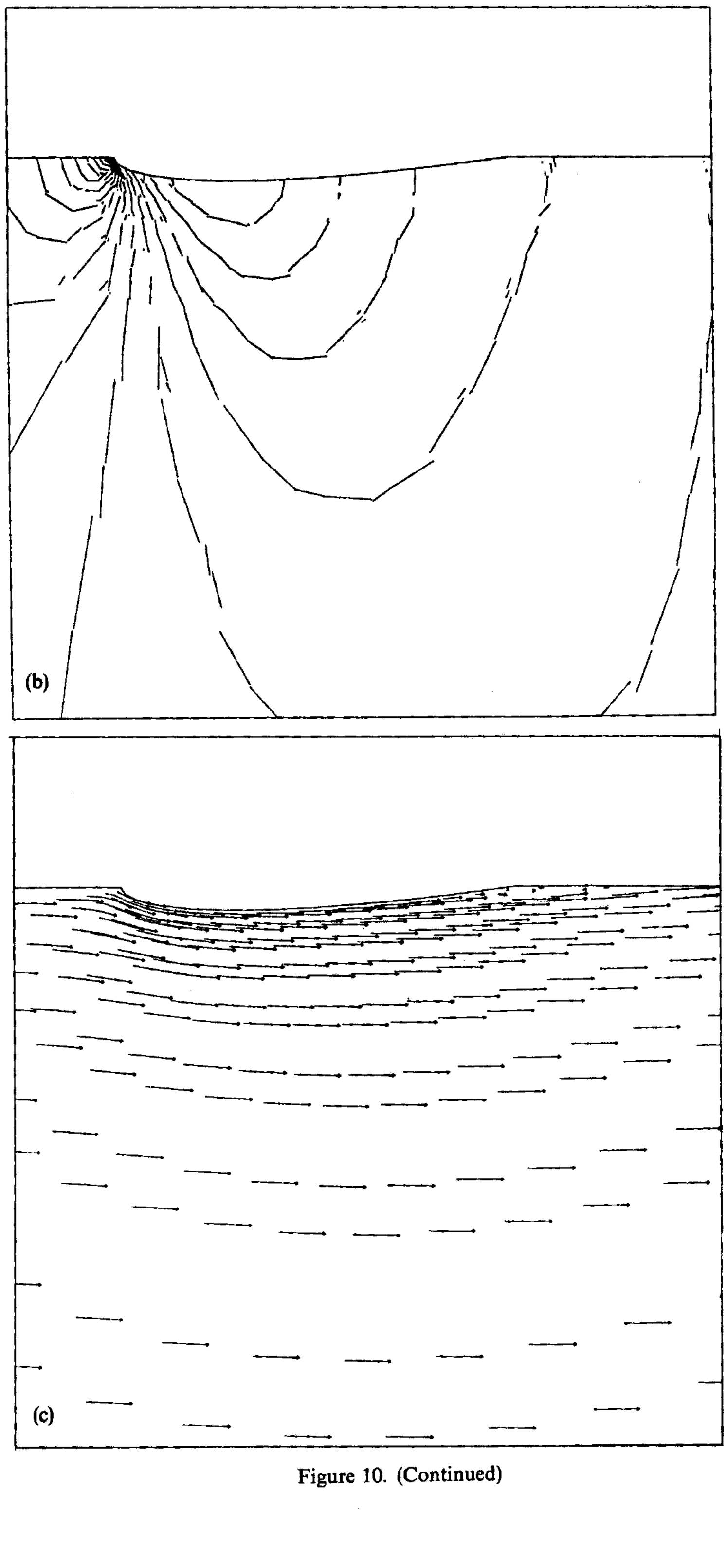

Figure 10. (Continued)

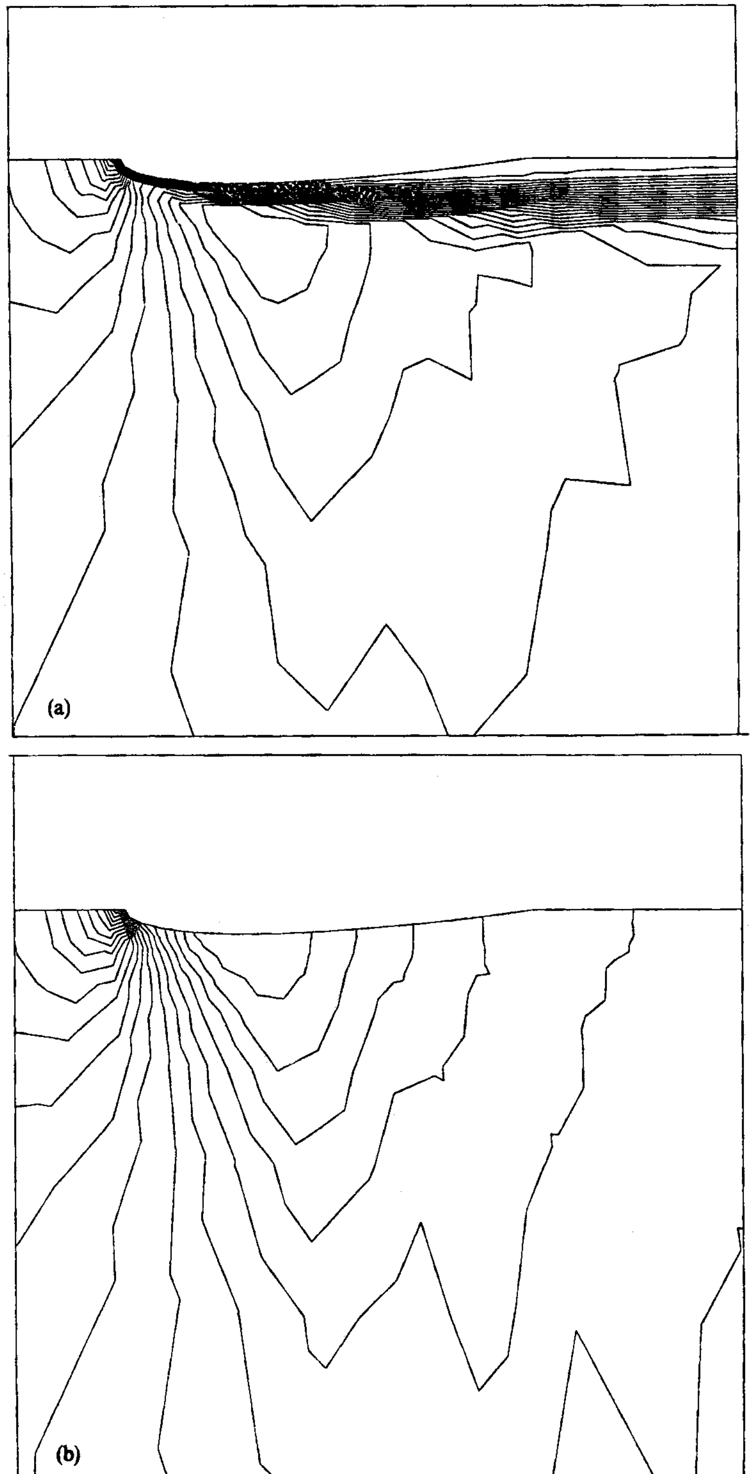

Figure 11. SUFL, perfect case: (a) isomach contours; (b) isobar contours; (c) stream vectors

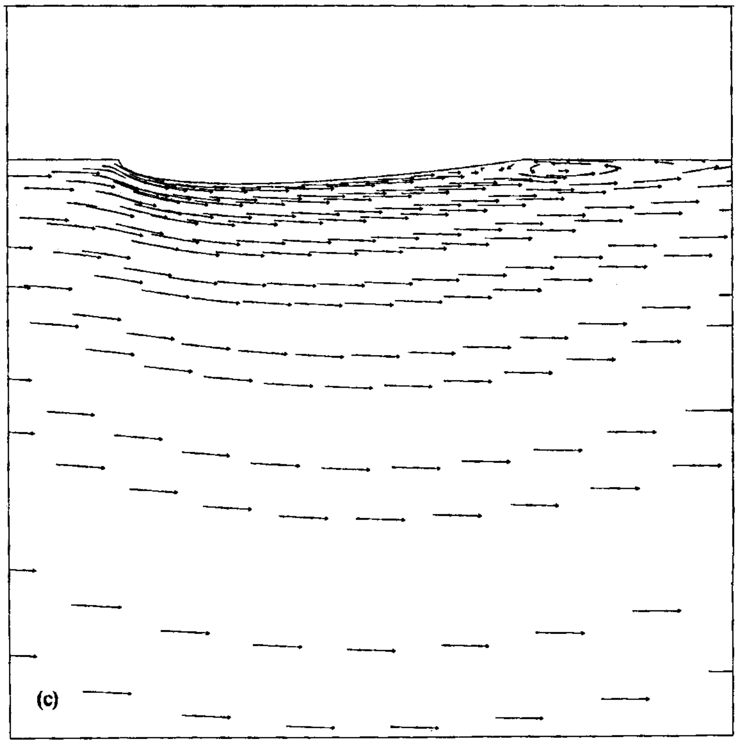

Figure 11. (Continued)

From these tests, we draw the following conclusions:

(a) Our smoothing method for discontinuous pressure approximations works effectively. However, at this point, we do not see a major advantage over simpler continuous approximations;
(b) Directional derivative methods are very diffusive as can be noticed by the absence of the vortex at the trailing edge and by a smaller maximum Mach number;
(c) SUFL method provides the less diffusive solution. However, it is not diffusive enough to handle high Reynolds number flows in presence of strong shocks. The variant of Section 4, behaved better, but some other shock-capturing mechanism is needed.

To check these conclusions, we performed further tests on the SUFL method using a perfect gas state law and a finer mesh (Figures 11(a)–11(c)) of Reynolds number varying from 500 to 10 000 past a NACA0012 aerofoil at $M=0.85$ and zero angle of attack. The solution exhibits a progression from steady solution to a periodic vortex shedding. Figures 12(a)–12(c) show comparisons of the pressure coefficient C_p results with those obtained by Cambier[15] and Boivin.[16] Good agreement is seen for $Re=10\,000$, the discrepancies are localized at the trailing edge which is more influenced by the unsteadiness of the flow. Figures 13(a), 13(b), 14(a)–14(c) show the mach, the pressure contours and the stream vectors for $Re=2000$ and $Re=10\,000$. It is clearly shown that the shocks and the boundary layers are very well resolved. Although the SUFL method effectively kills most oscillations, to accurately simulate flows characteristics we conclude that a minimal mesh size is required.

7. CONCLUSION

It was not our intent to present here realistic results of actual computations. Our purpose was to test non-standard methods for the discretization of incompressible flows. We have shown the feasibility of employing a discontinuous approximation for pressure and we have checked the large influence of upwinding techniques on the quality of results. A method introducing diffusion only where needed is still to be improved on the quality of results. Particularly a variant of Lesaint–Raviart scheme was developed to discretize advective term of discontinuous density field. Therefore, mass is conserved locally and there is a gain in computational time. A method introducing diffusion only where needed was investigated, and seems worth developing further.

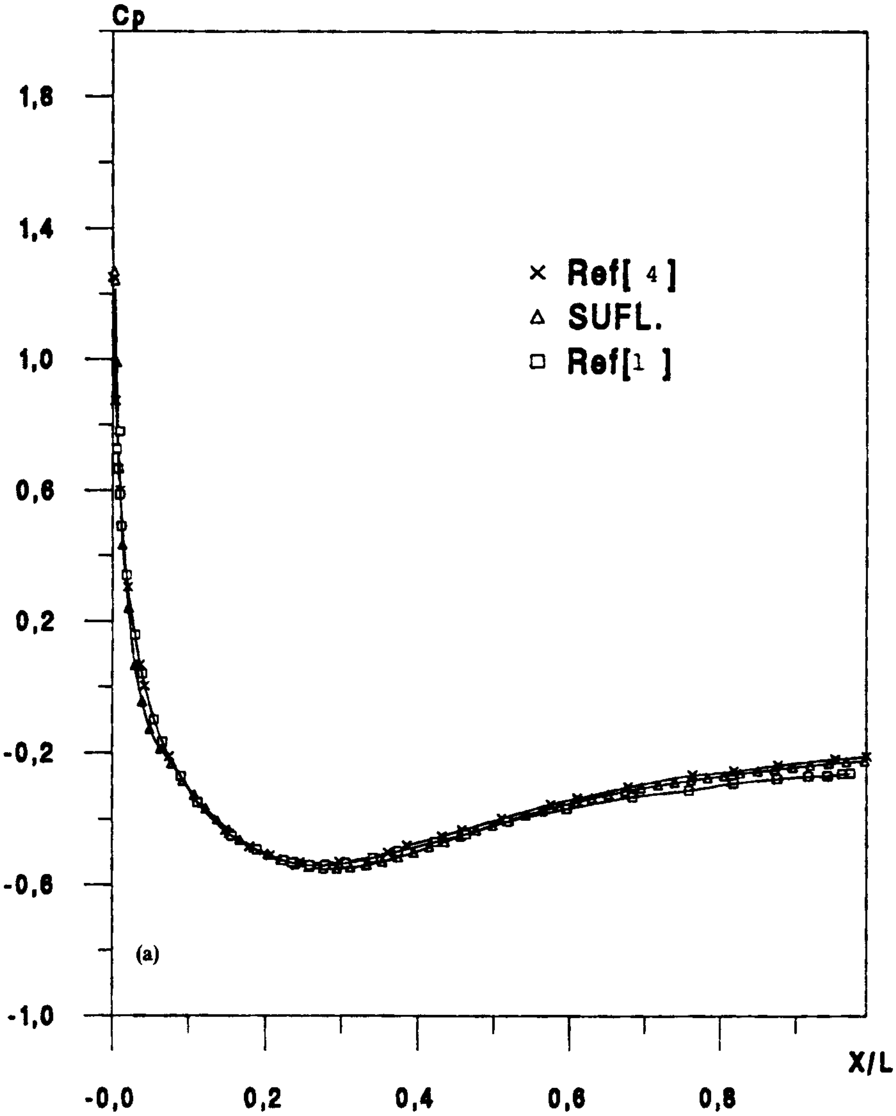

Figure 12. (a) C_p coefficients, $Re=2000$; (b) C_p coefficients at intrados, $Re=10\,000$; (c) C_p coefficients at extrados, $Re=10\,000$

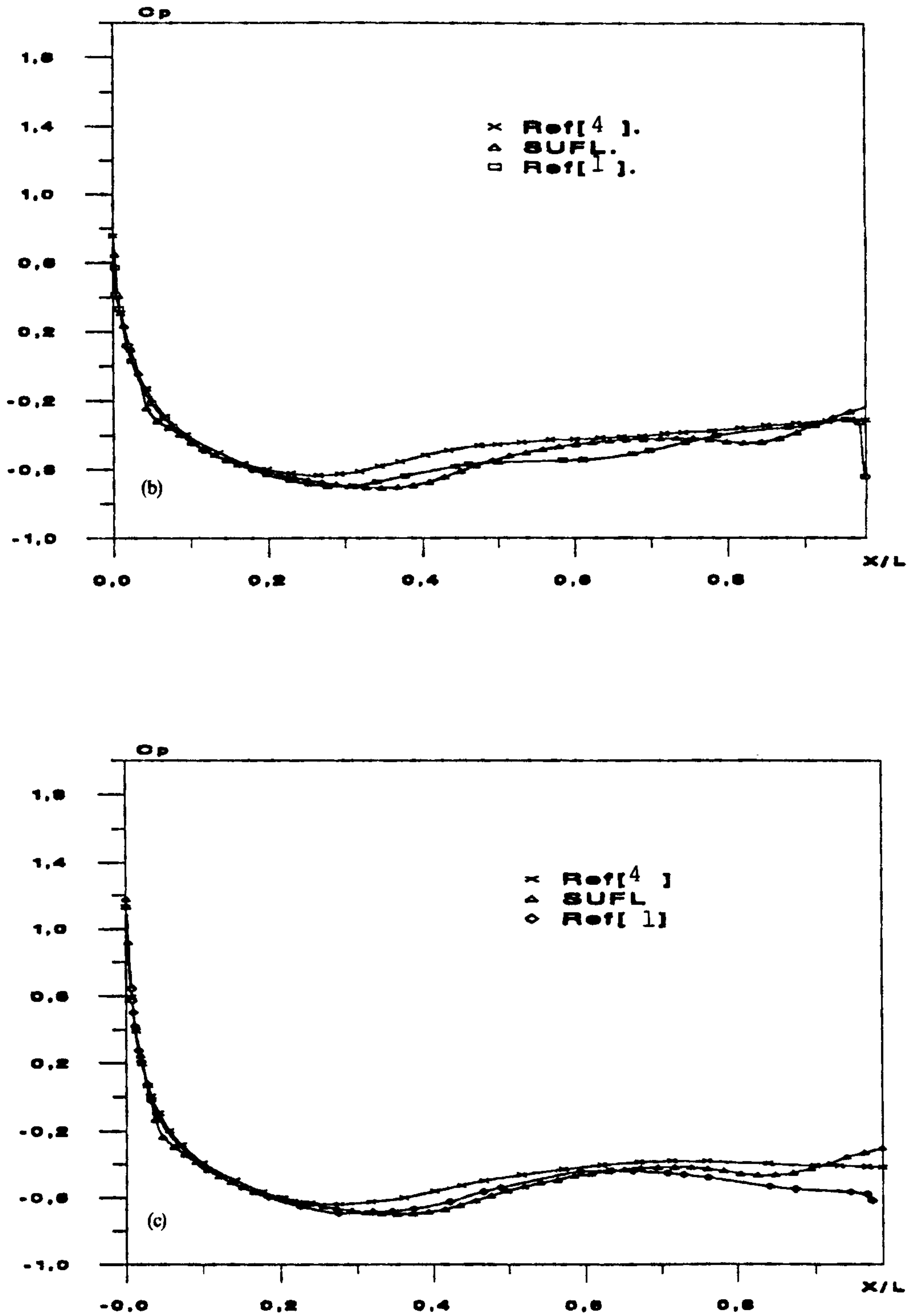

Figure 12. (Continued)

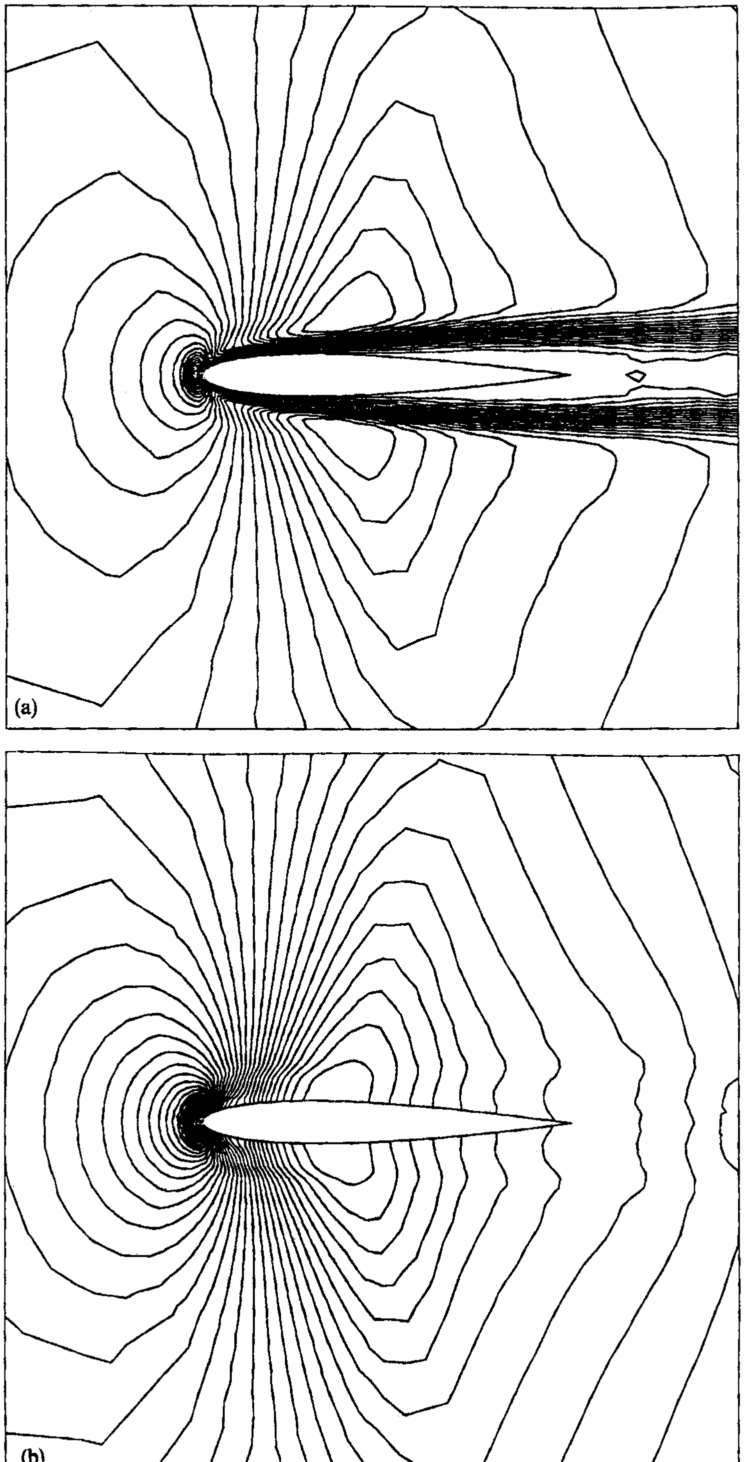

Figure 13. (a) Isomach contours, $Re=2000$, finer mesh; (b) Isobar contours, $Re=2000$, finer mesh

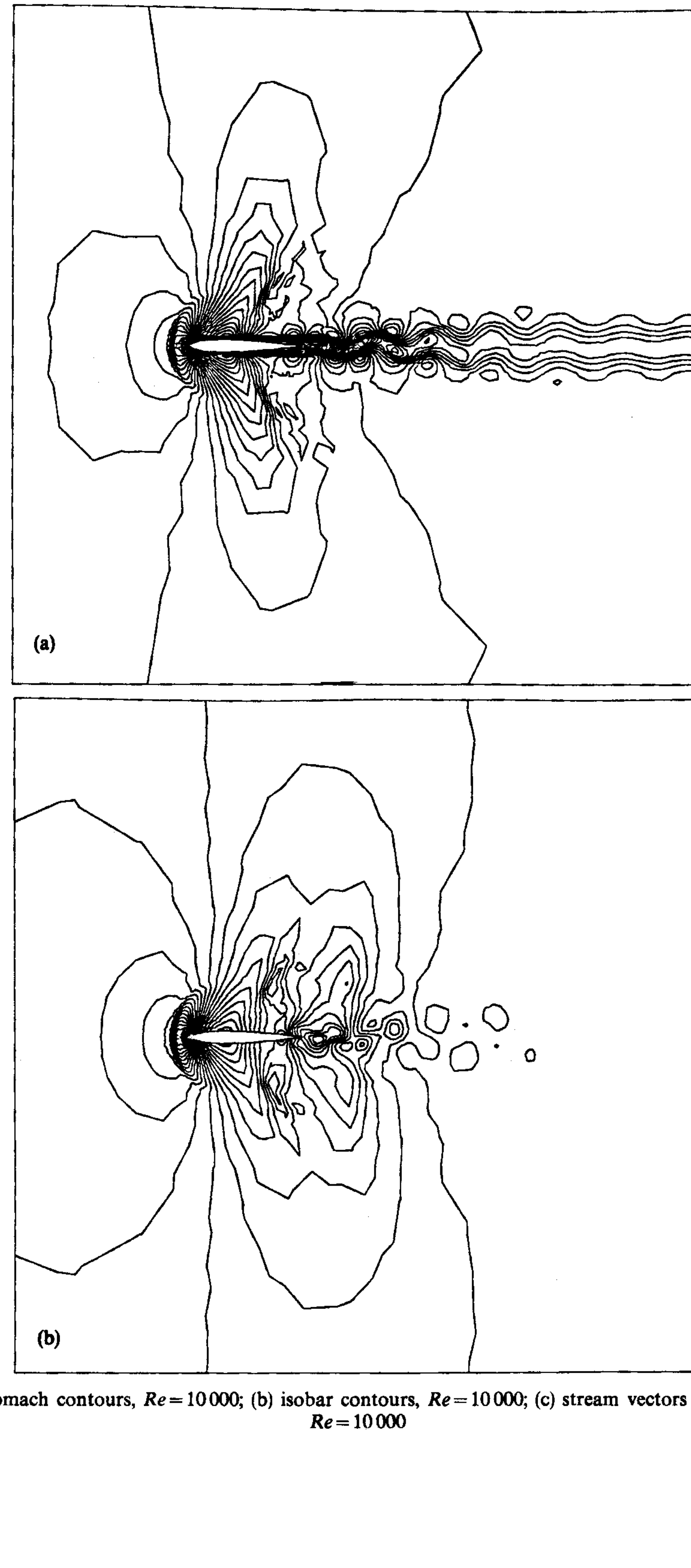

Figure 14. (a) Isomach contours, $Re = 10\,000$; (b) isobar contours, $Re = 10\,000$; (c) stream vectors at the leading edge, $Re = 10\,000$

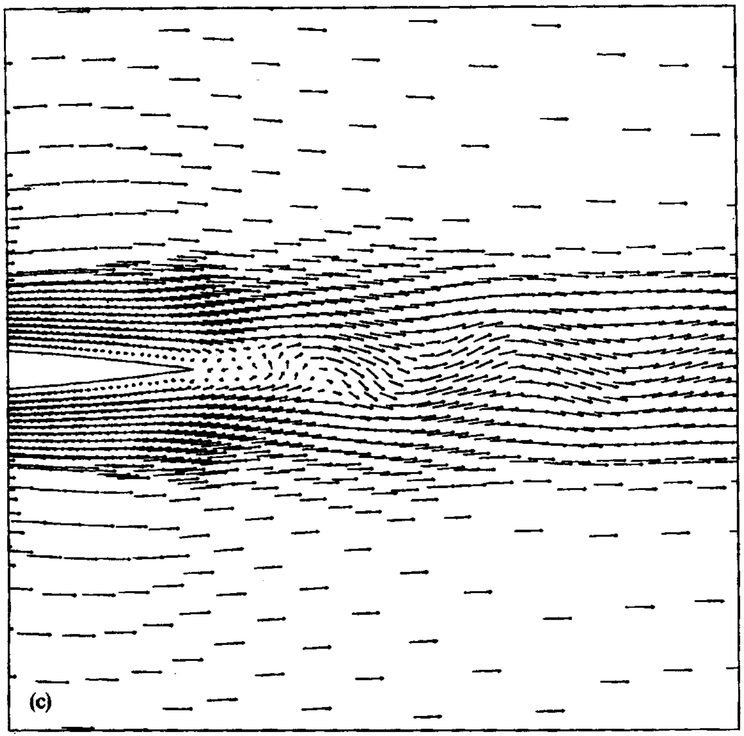

Figure 14. (Continued)

REFERENCES

1. M. Fortin and A. Soulaimani, 'Acceleration of compressible Navier–Stokes calculations', *Numerical Methods For Fluid Dynamics III*, Clarendon Press, Oxford, 1988.
2. F. Brezzi and M. Fortin, *Mixed and Hybrid Finite Element Methods*, Springer Series in Computational Mathematics, Springer, Berlin, 1991.
3. V. Girault and P. A. Raviart, *Finite Element Methods For Navier Stokes Equations*, Springer, Berlin, 1986.
4. T. J. R. Hughes, *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis*, Prentice-Hall, Englewood Cliffs, NJ, 1987.
5. C. Johnson, *Numerical Solution of Partial Differential Equations by the Finite Element Method*, Cambridge, 1987.
6. T. Hughes and A. Brooks, 'A multidimensional upwind scheme with no crosswind diffusion', *Finite Element Methods for Convection Dominated Flow*, ASME, New York, 1979, pp. 19–35.
7. T. J. R. Hughes and M. Mallet, 'A new finite element method for computational fluid dynamics III', *Comput. methods appl. mech. eng.*, **58**, (1986).
8. T. J. R. Hughes, M. Mallet and A. Mizukami, 'A new finite element method for computational fluid dynamics II', *Comput. methods appl. mech. eng.*, **54**, (1986).
9. M. Tabata and M. Yaoi, 'Upwind finite element approximation of the Navier–Stokes equation and its application to a free surface problem', *Proc. Int. Conf. on Computational Methods in Flow Analysis*, Okayama, 1988.
10. P. K. Sweeby, 'High resolution schemes using flux limiters for hyperbolic conservation laws', *SIAM J. Numer. Anal.*, **21**, 995–1011 (1984).
11. P. Lesaint and P. A. Raviart, 'On a finite element method for solving the neutron transport equation', *Mathematical Aspects of Finite Elements In Partial Differential Equations*, Academic Press, Cambridge, 1974.
12. Y. Saad and M. H. Schultz, 'A generalized minimal residual algorithm for solving nonsymmetric linear systems', *Research Report', YALEU-DCS-RR-254*, Yale University, New Haven, 1983.
13. L. B. Wigton, D. P. Yu and N. J. Young, 'GMRES acceleration of computational fluid dynamics codes', *Proc. 1985 AIAA Conf.*, Denver, 1985.
14. M. O. Bristeau, R. Glowinski, B. Mantel, J. Périaux and G. Rogé, 'Acceleration of compressible Navier–Stokes calculations', *Numerical Methods For Fluid Dynamics III*, Clarendon Press, Oxford, 1988.
15. L. Cambier, 'Computation of viscous transonic flows using an unsteady type method and a zonal grid refinement technique', O.N.E.R.A., France, 1985.
16. S. Boivin, 'Simulation d'écoulemets compressibles à nombre de Reynolds élevé', *Thèse de doctorat*, Université Laval, Canada, 1990