A Domain Decomposition Technique for Viscous/Inviscid Coupling

C.-H. LAI[1], A.M. CUFFE[2], & K.A. PERICLEOUS[3]

INTRODUCTION

The numerical simulation of viscous flows past realistic configurations using time-dependent compressible Navier-Stokes for high Reynolds numbers requires a large amount of computing time [cfd94]. Although steady-state solutions of a compressible inviscid flow past over a complete aircraft described by the Euler equation can be obtained in just a few minutes time, modelling 3-D unsteady flows to understand the aerodynamics of a complete aircraft is still a grand challenge. With the limits of computing resources, it is important to develop efficient numerical technqiues which are suitable for implementation on parallel computers and for engineering design purposes.

One such technique relies upon a problem partitioning concept. The concept in relation to viscous flows past over obstacles was originally observed by Prandtl [Pra04], of which the following two observations are involved. When a viscous fluid flow past a solid boundary, tangential and normal components of the velocity must both equal to the components of the velocity of the surface, and fluid sticks to the surface. For an ideal inviscid fluid, only the normal components are equal and there is no restriction on tangential slip of fluid relative to the boundary. The fundamental differences lead

[1] School of Computing & Mathematical Sciences, University of Greenwich, Wellington Street, Woolwich, London SE18 6PF, U.K. email: C.H.Lai@grn.ac.uk
[2] Address same as above. This author is supported by a Bursary of the University.
[3] Address same as above.

Domain Decomposition Methods in Sciences and Engineering, edited by R. Glowinski *et al.*

to the idea of a boundary layer first proposed by Prandtl. It is a thin layer of fluid adjacent to the solid surface within which viscous effects are dominant and that such effects become negligible outside the thin layer. Hence it is valid to divide the flow region into two parts, i.e. the boundary layer and the inviscid flow outside the layer.

Mathematically, one uses either the Navier-Stokes equation or one of its reduced form for flows within the boundary layer and an inviscid model described by either the Euler equation or the potential equation outside the boundary layer. Numerically, such problem partitioning is often referred to as the zonal method [Sch86]. There are two zones namely, inner and outer, with the inner zone being described by the thickness function t. The inner zone is governed by the Navier-Stokes equation and the outer zone is governed by the Euler equation. Due to the amount of computing power, one current industrial practice is to solve a reduced form of the Navier-Stokes equation in the boundary layer and then coupled with a compressible potential model outside the boundary layer. The coupling is obtained by means of an iterative technique similar to the one developed by Schwarz [Sch90] for elliptic problems.

One property of the zonal method is that a prior knowledge is required for the partitioning of the flow field. Hence there is a jump of mathematical models being used in different zones. It would be useful to have a dynamic zonal recognition capability with a smooth transition from one model to another model. It would also be useful to minimize the thickness function t such that accuracy obtained by means of the Navier-Stokes equation for viscous flow simulations can be retained.

In this paper, a summary is given of various truncation techniques, including the scalar truncation method used by Brezzi [BCR89]. An extension is given of the scalar truncation method to the Navier-Stokes equation in the context of a finite volume method in order to build a dynamic zonal recognition capability into an in-house cell-centred finite volume multiphase software UIFS [Cho93].

TRUNCATION TECHNIQUES

Truncation methods have been around for a long time. It has been used, in a different form, for the determination of local hyperbolic and elliptic regions in a plane transonic flows for shock location [MC71] or local adaptive mesh refinement for shock capturing. This Section describes two frequently used truncation techniques related to flows past an obstacle.

Prandtl's truncation

Prandtl introduced the idea of boundary layer by neglecting the effect of viscosity outside the thin layer next to a surface. One assumption to implement the idea in 2-D incompressible viscous flows past an obstacle described by

$$\nabla \cdot \underline{u} = 0 \tag{1}$$

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\underline{u}\phi - \nu \nabla \phi) = -\frac{1}{\rho}\frac{\partial p}{\partial \underline{n}} \tag{2}$$

where $\phi = u, v$ and $\underline{u} = (u, v)^T$ is $u \gg v$. The assumption leads to the boundary layer equations as described by

$$\nabla \cdot \underline{u} = 0 \tag{3}$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\frac{\partial^2 u}{\partial y^2} \tag{4}$$

$$\frac{1}{\rho}\frac{\partial p}{\partial y} \approx 0 \tag{5}$$

There are other assumptions which reduces the 2-D incompressible viscous flow model to other turbulent boundary layer models [Kno86] or models suitable for regions with separations and wakes [GWB73] [Rau75].

Scalar truncation

Early work involving the concept of a scalar truncation technique can be found in Perkins and Rodrigue paper related to a dynamic zonal recognition process for viscous Burgers' equation [PR89]. The idea is to implement in a finite difference context rather than the manual truncation technique as used by Prandtl. Finite difference representations of the viscous term at various grid points across the whole flow domain are calculated, a viscous region across the flow domain can then be masked out leaving the rest of the flow domain consists of an inviscid region at least numerically. One disadvantage of this method is that the viscous term is calculated across the whole computational domain which is extremely ineffective. Also the method does not provide a smooth transition from one mathematical model to the other, and hence the solution is not necessarily smooth across the boundary of the models. An improved version was introduced by Brezzi [BCR89] such that the transition of the viscous term from the viscous region to the inviscid region is smooth.

The zonal recognition is based on the scalar truncation method,

$$\mathcal{T}(q) = \begin{cases} 0, & |q| \le \epsilon \\ f(q), & \epsilon < |q| < \sigma \\ q, & |q| \ge \epsilon + \sigma \end{cases} \tag{6}$$

where q is certain physical quantities and ϵ and σ are two threshold parameters which can be adjusted to produce numerical results close enough to the true solution. For the present study q represents the viscous term. Work by Perkins and Rodrigue was to choose $\sigma = 0$ in the above scalar truncation method. Brezzi chose $f(q)$ as a straigt line and Canuto chose $f(q)$ as a cubic spline [AC93]. The main concern here is that a smooth transition of the viscous term should be maintained. Therefore, it is equivalent to the determination of the pair of parameters, ϵ and σ, such that the solution to the Burgers equation is most accurate under certain error norms. Some comparisons of results using various threshold parameters can be found in [AC93][BCR89][PR89]. Since $\mathcal{T}$ is nonlinear which means in general, $\mathcal{T}(\frac{\partial^2 u}{\partial x^2}) \neq \frac{\partial}{\partial x}(\mathcal{T}(\frac{\partial u}{\partial x}))$, and hence it is only suitable in the context of finite difference methods. An extension, given in the following Section, to two-dimensional problems which involves finite volume methods requires different treatment. Similar extensions to finite element methods can be found in [AP93].

A Vector Truncation Method

For two-dimensional incompressible viscous flow, the momentum equation given in (2) is integrated over a control volume Ω using a finite volume technique, i.e.

$$\int\int_{\Omega} \frac{\partial \phi}{\partial t} d\Omega + \int\int_{\Omega} \nabla \cdot (\underline{u}\phi - \nu \nabla \phi) d\Omega = -\frac{1}{\rho} \int\int_{\Omega} \frac{\partial p}{\partial \underline{n}} d\Omega \tag{7}$$

which is re-written as

$$\int\int_{\Omega} \frac{\partial \phi}{\partial t} d\Omega + \int_{\partial\Omega} (\underline{u}\phi - \nu \nabla \phi) \cdot \underline{n} ds = -\frac{1}{\rho} \int\int_{\Omega} \frac{\partial p}{\partial \underline{n}} d\Omega \tag{8}$$

where $\underline{n}$ denotes the unit normal vector. The fluid flow inside a viscous region is certainly a rotational flow, and hence velocity gradients are the major contribution to viscous effect. Therefore it is reasonable to apply truncations to these velocity gradients, i.e.

$$\int\int_{\Omega} \frac{\partial \phi}{\partial t} d\Omega + \int_{\partial\Omega} (\underline{u}\phi - \nu \underline{\mathcal{T}}(\nabla \phi)) \cdot \underline{n} ds = -\frac{1}{\rho} \int\int_{\Omega} \frac{\partial p}{\partial \underline{n}} d\Omega \tag{9}$$

should replaces (8), and $\underline{\mathcal{T}}$ denotes the vector truncation methods,

$$\underline{\mathcal{T}}(\nabla \phi) \ = \ (\mathcal{T}(\frac{\partial \phi}{\partial x}) \ , \ \mathcal{T}(\frac{\partial \phi}{\partial y}))^{\mathrm{T}} \tag{10}$$

Since $\frac{\partial u}{\partial y}$ and $\frac{\partial v}{\partial x}$ contribute to the rotational effect of the flow in the viscous region, therefore these two velocity gradients are significant compare with $\frac{\partial u}{\partial x}$ and $\frac{\partial v}{\partial y}$. Hence $\frac{\partial u}{\partial x}$ and $\frac{\partial v}{\partial y}$ reduce to a smaller value earlier than $\frac{\partial u}{\partial y}$ and $\frac{\partial v}{\partial x}$ with the application of (10). Thererfore (9) effectively reduces the Navier-Stokes equation to a truncated integral form of Navier-Stokes near the outer zone but remains inside the boundary layer and to the Euler equation further away from the body.

The truncation method is embedded into the finite volume formulation which ensures that the boundary layer is evolved as part of the numerical solution. It allows a smooth transition of mathematical models from the Navier-Stokes to the Euler equation.

NUMERICAL RESULTS

First, numerial tests is provided for a flat plate problem. Instead of using the infinite upper half plane, the finite domain $\{(x, y) : -0.5 \leq x \leq 2 \ , \ 0 \leq y \leq 10\}$ is used, where the flat plate is placed in $0 \leq x \leq 1$. There are 50×50 cells along x and y-axis, $1/5$ of the cells along y-axis are located in $0 \leq y \leq 0.2$ and the rest are located in $0.2 < y \leq 10$. The upstream Reynolds number is chosen to be 1000. The vector truncation method is tested with a series of ϵ and σ parameters. The results are compared with the Blasius solution, with the package Phoenics and with the in-house cell-centred finite volume software UIFS [Cho93] and are presented in Figure 1. Figure

2 shows the effect of $||u_{\epsilon,\sigma} - u_{fv}||_\infty$ for different values of ϵ and σ used in the flat plate problem, where $u_{\epsilon,\sigma}$ denotes the numerical solution of u by taking ϵ and σ as the threshold parameters and u_{fv} denotes the numerical solution of u by using the software UIFS.

Second, numerical tests are provided for the NACA0012 aerofoil at zero angle of incidence. Therefore only part of the upper half of the physical domain is considered and a C-type of mesh is used for subsequent computation. The finite physical domain is mapped onto the computational domain $\{(\xi, \eta) : 0 \le \xi \le 2\ ,\ 0 \le \eta \le 10$ where the transformed aerofoil is a straight line located in $0 \le \xi \le 1$. There are 40×100 cells along ξ and η-axis, 1/5 of the cells along η-axis are located in $0 \le \eta \le 0.2$ and the rest are located in $0.2 \le \eta \le 10$. There are 25 cells along the aerofoil surface. The upstream Reynolds number is chosen to be 1000. Figure 3 shows the velocity profiles against the distance along η-axis, obtained at $\xi = 0.46$ and $\xi = 0.66$. Figure 4 shows the effect of $||u_{\epsilon,\sigma} - u_{fv}||_\infty$ for the aerfoil problem.

In general, the finite volume solution obtained by means of applying the vector truncation technique to the Navier-Stokes equation is close enough, in the sense of engineering purpose, compare to the corresponding finite volume solution obtained by means of the Navier-Stokes equation. From Figures 2 and 4, the solution accuracy of the flat plate problem and the NACA0012 aerofoil probelm is not sensitive to the choice of σ.

CONCLUSIONS

The scalar truncation method is extended to a vector truncation method and is embedded in a cell-centred finite volume method. The effect of the threshold parameter σ is insignificant to the accuracy of the solution. The choice of the threshold parameters namely, σ and ϵ, is effectively equivalent to the determination of a minimal inner region defined by the thickness function t. The present approach shows that the computation of velocity gradients in the finite volume method can be used in both the evaluation of the line integrals as well as for viscous effect comparison.

REFERENCES

[AC93] Arina R. and Canuto C. (1993) A self-adaptive domain decomposition for the viscous/inviscid coupling i: Burgers equations. *J. of Computational Physics* 105: 290–300.

[AP93] Achdou Y. and Pironneau O. (1993) The χ-method for the Navier-Stokes equations. *IMA J. of Numer. Analysis* 13: 537–558.

[BCR89] Brezzi F., Canuto C., and Russo A. (1989) A self-adaptive formulation for the Euler/Navier-Stokes coupling. *Computer Methods in Applied Mechanics and Engineering* 73: 317–330.

[cfd94] (1994) *New Opportunities and Directions in Aeronautical CFD*, number 16 in EASE Programme: CFD Community Club Seminars, Chilton, Didcot, UK.

[Cho93] Chow P. M.-Y. (1993) *Control Volume Unstructured Mesh Procedure for Convection-Diffusion Solidification Process.* PhD dissertation, University of Greenwich.

[GWB73] Green J., Weeks D., and Brooman J. (1973) Prediction of turbulent boundary layers and wakes in incompressible flow by a lag entrainment method. Technical Report and Memo T3792, Royal Aircraft Establishment, Farnborough, UK.

[Kno86] Knott M. (1986) *Numerical Solutions of Two-Dimensional Unsteady Boundary Layers.* PhD dissertation, University of London.

[MC71] Murman E. and Cole J. (1971) Calculation of plane steady transonic flows. *AIAA J.* 9: 114–121.

[PR89] Perkins A. and Rodrigue G. (1989) A domain decomposition method for solving a two-dimensional viscous burgers' equation. *Applied Numerical Mathematics* 6: 329–340.

[Pra04] Prandtl L. (1904) Über Flüssigkeitsbewegung bei sehr kleiner reibung. In *Proceedings of the 3rd International Mathmatics Congress.* Heidelberg.

[Rau75] Raudkivi A.J. & Callander R. (1975) *Advanced Fluid Mechanics.* Edward Arnold (Publisher) Ltd, London.

[Sch90] Schwarz H. (1890) Über einen Grenzübergang durch alternierendes verfahren. *Gesammelte Mathematische Abhandlungen* 2: 133–143.

[Sch86] Schmatz M. (1986) Calculation of strong interactions on airfoils by zonal solutions of the Navier-Stokes equations. In Rues D. and Kordulla W. (eds) *Notes on Numerical Methods in Fluid Mechanics*, number 13, pages 335–342. Vieweg, Braunschweig-Wiesbaden.

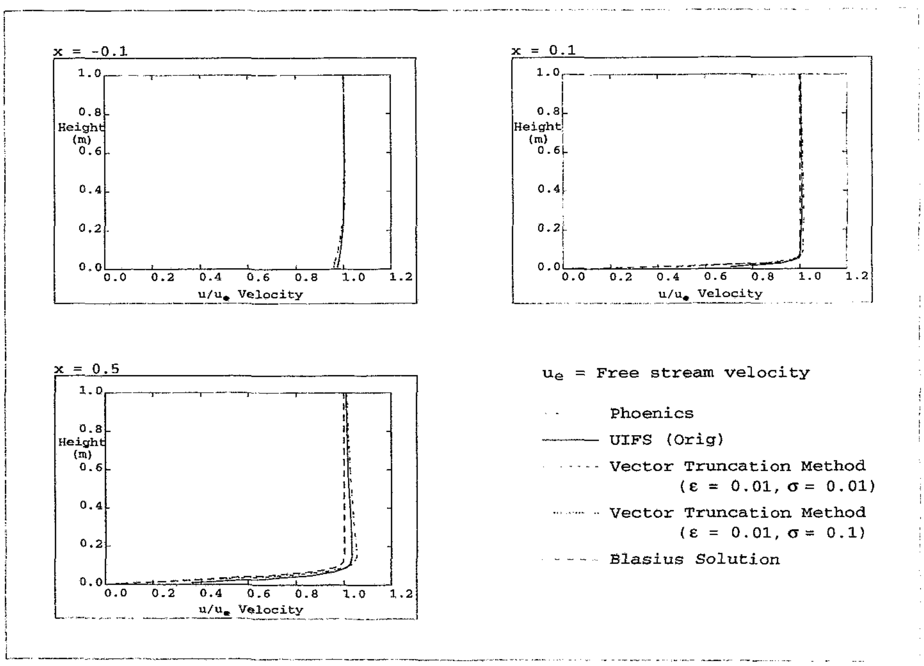

Figure 1 The flat plate problem.

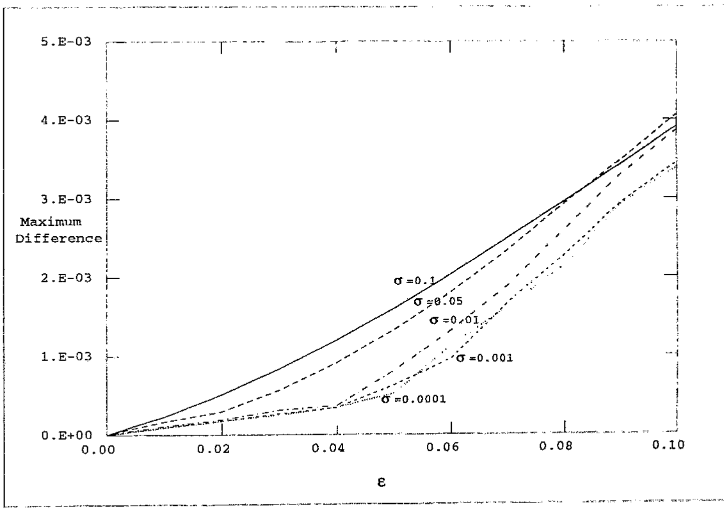

Figure 2 $||u_{\epsilon,\sigma} - u_{fv}||_\infty$ for some values of ϵ and σ.

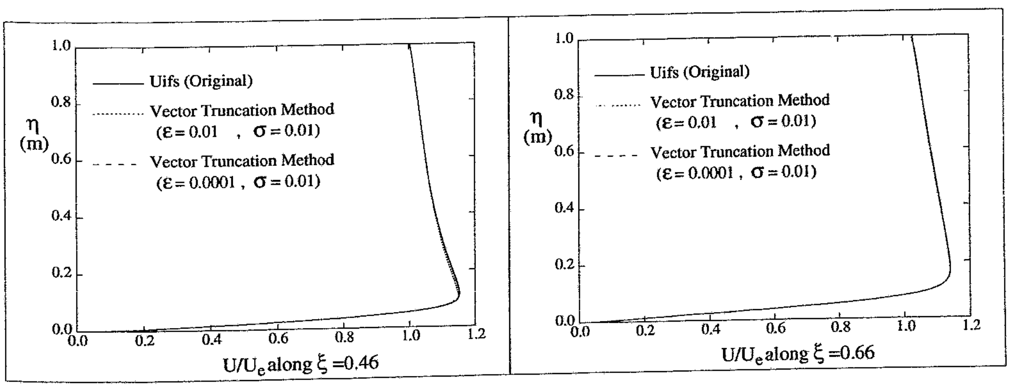

Figure 3 Velocity profile along $\xi = 0.46$ and $\xi = 0.66$.

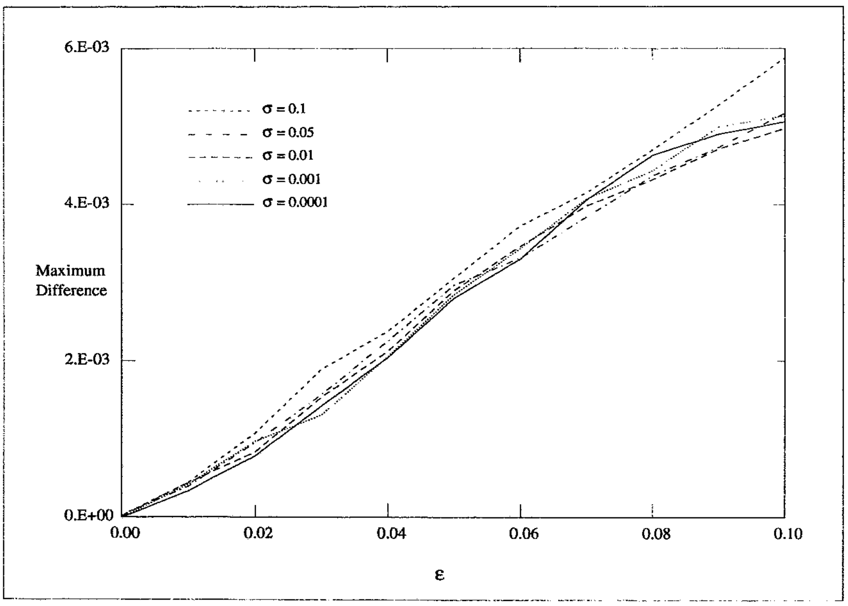

Figure 4 $||u_{\epsilon,\sigma} - u_{fv}||_\infty$ for some values of ϵ and σ.