81

Adaptive Zonal Recognition for Viscous/Inviscid Coupling

A. M. Cuffe, C.-H. Lai, and K. A. Pericleous

1 Introduction

The Navier-Stokes equation is the model generally used to describe the flow of a viscous fluid. In its incompressible form this model is written as

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\underline{u}\phi) = \nu \nabla^2 \phi - \frac{1}{\rho}\nabla p, \tag{1.1}$$

$$\nabla \cdot \underline{u} = 0, \tag{1.2}$$

where $\phi = u, v$ and $\underline{u} = (u, v)^T$.

This system can be computationally demanding. As is well known, high Reynolds number laminar flows produce a boundary layer that is thin compared to the overall flow domain. As a result, time may be wasted by solving the Navier-Stokes equations across the whole domain when the Euler equations would be more appropriate in a large proportion of the domain.

A solution is to divide the flow into viscous and inviscid regions and solve these two regions separately using a suitable iterative technique. This is known as viscous/inviscid interaction [Kno86], and requires zones to be manually predefined [SH86].

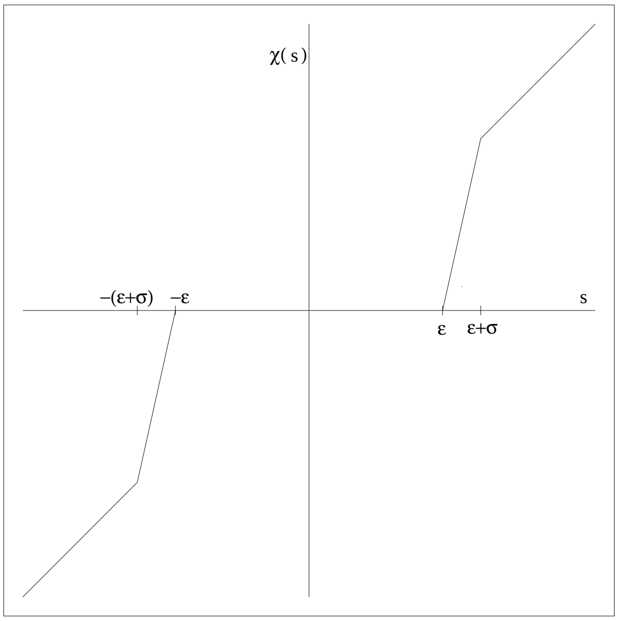
This paper describes the development of an adaptive zonal recognition procedure that does not require manual partitioning and therefore overcomes many of the disadvantages of early zonal methods.

2 Zonal Recognition

The correct choice of zonal boundaries is very important in viscous/inviscid coupling in order to achieve an efficient solution. Consequently there is considerable interest in developing zonal recognition techniques. Work by Perkins and Rodrigue in 1989 [PR89]

Ninth International Conference on Domain Decomposition Methods
Editor Petter E. Bjørstad, Magne S. Espedal and David E. Keyes

Figure 1 The χ function with a straight line in the transition region



involved computing finite difference value of the viscous term at discrete points. In the same year Brezzi, Canuto and Russo [BCR89] developed a zonal recognition function called the χ-method. More recently Margot [Mar93] developed a physically guided zonal approach. This involved running an initial course grid problem and examining the magnitude of the viscous terms in order to give an indication of the best position for the zonal boundaries. Current work by the authors aims to develop an adaptive domain decomposition technique with zonal recognition and decoupling within a single code framework.

The development of the zonal recognition technique is based on the χ-method by Brezzi, Canuto and Russo. The χ-method can be considered as a truncation technique that reduces the Navier-Stokes system to the Euler in regions where the viscous term is small. This is done by replacing the viscous term $\nabla^2\phi$ in the Navier-Stokes equation by a function $\chi(\nabla^2\phi)$. This function coincides with $\nabla^2\phi$ when the viscous term is large and equates to zero when its value is small, thus becoming the Euler equation.

In particular, the χ function described by Brezzi may be written,

$$\chi(s) \;=\; \begin{cases} 0, & |s| \leq \epsilon \\ f(s), & \epsilon < |s| < \epsilon+\sigma \\ s, & |s| \geq \epsilon+\sigma \end{cases} \tag{2.3}$$

where s is, in this case, the viscous term. The values ϵ and σ are threshold parameters which define the size of the viscous and transition regions respectively. By taking a strictly increasing function in the transition region between ϵ and $\epsilon+\sigma$ the χ function becomes a monotonically increasing, continuous function. Brezzi chose a straight line in this region (see Figure 1), while Arina and Canuto used a third-degree polynomial [AC93].

Brezzi originally applied the χ-method in a finite difference context and preliminary results for a one-dimensional test problem have shown that this method works well using a finite difference discretisation. More recently, Achdou and Pironneau [AP93] have applied the χ-method in a finite element method. We have incorporated the χ-method in a finite volume context [LCP96].

3 A Truncation Technique for Finite Volume Methods

Solving the Navier-Stokes equation using a finite volume method involves the integration over a control volume Ω. The Navier-Stokes equation then becomes,

$$\int\int_{\Omega} \frac{\partial \phi}{\partial t}\, d\Omega + \int\int_{\Omega} \nabla \cdot (\underline{u}\phi)\, d\Omega = \int\int_{\Omega} \nu \nabla^2 \phi\, d\Omega - \int\int_{\Omega} \frac{1}{\rho} \nabla p\, d\Omega, \tag{3.4}$$

which can be rewritten as

$$\int\int_{\Omega} \frac{\partial \phi}{\partial t}\, d\Omega + \int_{\partial\Omega} (\underline{u}\phi) \cdot \underline{n}\, ds = \int_{\partial\Omega} \nu (\nabla \phi \cdot \underline{n})\, ds - \int\int_{\Omega} \frac{1}{\rho} \nabla p\, d\Omega. \tag{3.5}$$

It is difficult to truncate the diffusion term in this form as it is now represented by a surface integral. Therefore a modification to the original truncation method is required. The contribution to viscous effect comes from the shear stress at the cell faces, therefore it is reasonable to apply the truncation method to the velocity gradients. Thus, the equation becomes

$$\int\int_{\Omega} \frac{\partial \phi}{\partial t}\, d\Omega + \int_{\partial\Omega} (\underline{u}\phi) \cdot \underline{n}\, ds = \int_{\partial\Omega} \nu\, \underline{\chi}(\nabla \phi) \cdot \underline{n}\, ds - \int\int_{\Omega} \frac{1}{\rho} \nabla p\, d\Omega \tag{3.6}$$

where $\underline{n}$ denotes the unit normal vector, and the vector truncation method is denoted by

$$\underline{\chi}(\nabla \phi) \;=\; (\chi(\frac{\partial \phi}{\partial x})\;,\; \chi(\frac{\partial \phi}{\partial y}))^T. \tag{3.7}$$

The vector truncation method allows a smooth transition of mathematical models from Navier-Stokes to parabolised Navier-Stokes and then to Euler.

This method has been successfully implemented in an in-house two-dimensional finite volume Navier-Stokes code that uses a cell-centred approach. Early studies have shown that the the vector truncation method gives numerical results similar to those obtained by using the finite volume method, the error being dependent on the choice of parameters ϵ and σ. By adjusting these parameters a solution can be obtained which is closer to the finite volume solution. A number of numerical tests have been performed with varying values for the two parameters ϵ and σ on a flat plate problem and an aerofoil problem [LCP96].

From these tests, it has become apparent that keeping the value of ϵ small reduces the error $\|u_\chi - u_{fv}\|_\infty$, where u_{fv} is the numerical solution obtained by the finite volume method, and u_χ is the numerical solution from the truncation method. This means that as the size of the inviscid region is reduced, the error between the vector truncation method and the finite volume method is also reduced. Varying the value of σ, i.e. the size of the transition region, also has an effect on the error. Again it is observed that as the value of σ is increased the error also increases. However the error due to increasing σ does not appear to be as significant, and it depends to a certain extent on the chosen value of ϵ. For example, large ϵ together with large σ will produce a large error due to the viscous region being very small. However if ϵ is small then the effect of having a large σ is greatly reduced. It is clear that careful consideration is required if the best combination for these two parameters is to be chosen.

4 Navier-Stokes/Euler Coupling

With the vector truncation method in place it is possible to use this truncation technique as the basis for an adaptive zonal recognition procedure. The finite volume code has been further developed so that once identified the regions are decoupled and solved separately.

An Euler code has been incorporated into this single code framework to solve the large inviscid region. The Euler equations are written as,

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0, \tag{4.8}$$

where

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \end{pmatrix}, \qquad F = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \end{pmatrix}, \qquad G = \begin{pmatrix} \rho v \\ \rho v u \\ \rho v^2 + p \end{pmatrix}. \tag{4.9}$$

For incompressible flow the energy equation can be replaced by the condition of constant total enthalpy

$$\frac{\gamma}{\gamma - 1}\frac{p}{\rho} + \frac{1}{2}(u^2 + v^2) = H_\infty. \tag{4.10}$$

The discretised Euler equations are rearranged so that grid points along a vertical column of the body fitted coordinate form a tridiagonal system. The whole region is solved column by column using a tridiagonal solver, which corresponds to a semi-implicit scheme. The algorithm is easy to implement and is sufficient at this stage to test the coupling procedure.

The algorithm for this single framework is shown in Figure 2. A finite volume Navier-Stokes code is run for a number of sweeps so that the solution develops just enough for the zonal boundaries to be identified. Decoupling then occurs with the initial solution being mapped onto the two subdomains. The existing finite volume Navier-Stokes code is coupled with a finite volume Euler code. Suitable Neumann and Dirichlet boundary conditions are imposed for the purpose of exchanging information between the two subdomains.

5 Numerical Results

Numerical tests have been carried out on a 1m flat plate problem with threshold parameters initially set at $\epsilon = 0.01$ and $\sigma = 0.0001$ (see Fig. 3). These results are promising although the coupling between viscous and inviscid regions still requires some adjustments. The internal interface between the two regions uses Dirichlet/Neumann boundary conditions for Navier-Stokes and Euler regions respectively. Other combinations have been tried but this appears to give the best results. The discrepancy in the Euler solution close to the viscous/inviscid interface is being investigated. This may be caused by the way in which the boundary conditions are applied.

6 Discussion and Conclusions

Viscous and inviscid regions of a flow domain can be identified by means of a vector truncation method within a finite volume Navier-Stokes framework. The regions are decoupled and solved iteratively within a single code. This is a great advantage as it saves computational time and overcomes the problems of more traditional coupling methods. Zones no longer need to be predefined and there is no need to choose boundary conditions for the overlapping zonal boundaries. Boundary values can be taken from the developing solution.

For greater computational speed the Euler/Navier-Stokes coupling may also be thought of as two codes working within a single framework. This enables the two codes to be run simultaneously on two processors with boundary information being exchanged between processors at regular intervals.

Increased accuracy in the viscous region is an important aspect. Greater accuracy in the boundary layer could be achieved by adapting the mesh in this region. The truncation method will be used for zonal recognition and mesh points will be drawn from the inviscid region into the viscous region. In this way greater accuracy can be obtained where the physics is most interesting without having to add points.

REFERENCES

[AC93] Arina R. and Canuto C. (1993) A self-adaptive domain decomposition for the viscous/inviscid coupling. I. Burgers equation. *Journal of Computional Physics* 105: 290+.

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

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

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

[LCP96] Lai C. H., Cuffe A. M., and Pericleous K. A. (1996) A domain decomposition algorithm for viscous/inviscid coupling. *Advances in Engineering Software* 26: 151+.

[Mar93] Margot X. M. (August 1993) *A Physically Guided Zonal Approach for Euler/Navier-Stokes Predictions of Aerofoil Flows*. PhD thesis, University of

London, Department of Mechanical Engineering, Imperial College.

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

[SH86] Schmatz M. A. and Hirschel E. H. (1986) Zonal solutions for airfoils using Euler, boundary-layer and Navier-Stokes equations. In *Applications of Computional Fluid Dynamics in Aeronautics, AGARD Conference proceedings, 412*.

Figure 2 Flow diagram

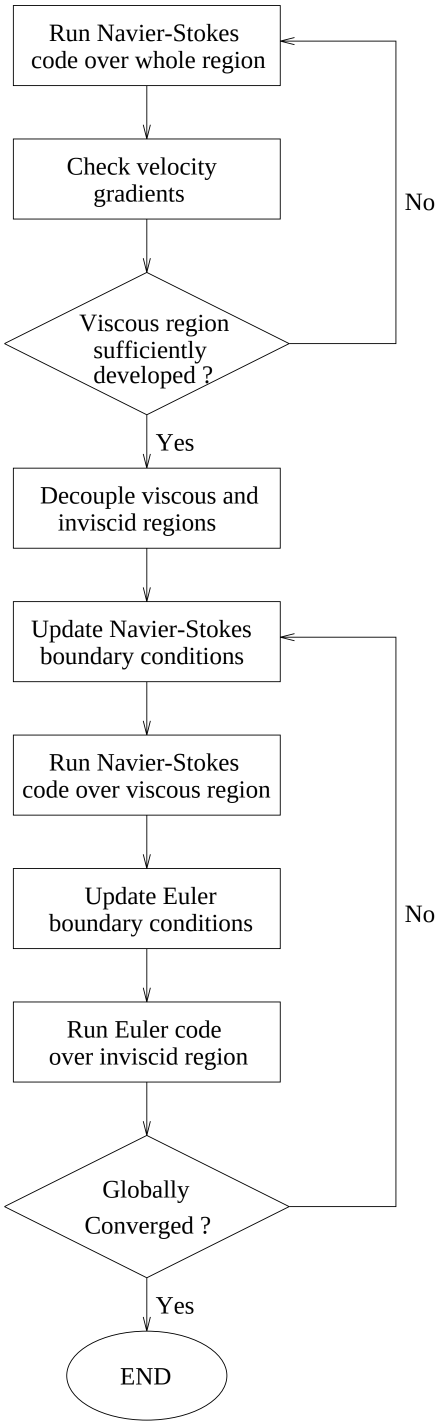


Figure 3 Graph to show velocity profiles at 3 points along a 1m Flat Plate