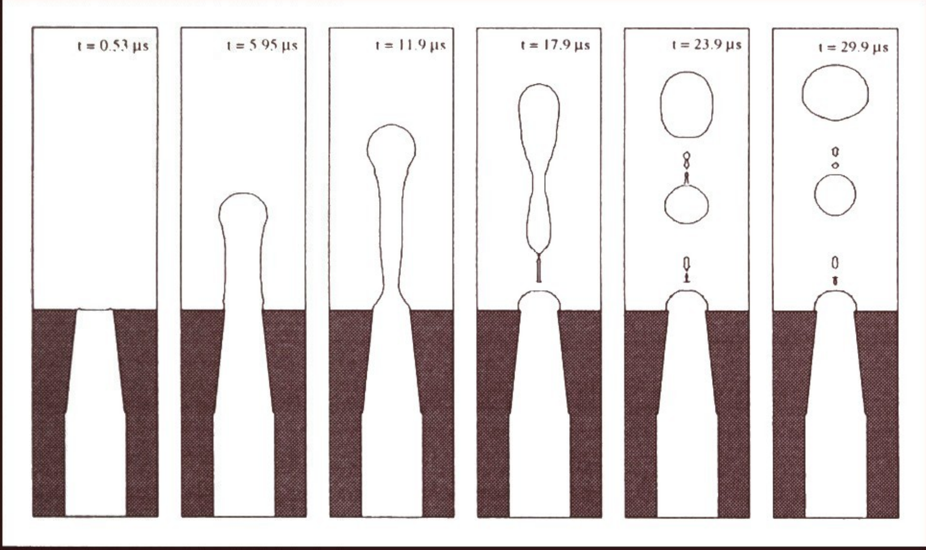
ICASE/LaRC Interdisciplinary Series in Science and Engineering

Barriers and Challenges in Computational Fluid Dynamics



Edited by
V. Venkatakrishnan, Manuel D. Salas
and Sukumar R. Chakravarthy

Springer-Science+Business Media, B.V.

Barriers and Challenges in Computational Fluid Dynamics

ICASE/LaRC Interdisciplinary Series in Science and Engineering

Volume 6

Barriers and Challenges in Computational Fluid Dynamics

edited by

V. Venkatakrishnan

Boeing Commercial Airplane Group,
Seattle, Washington, U.S.A.

Manuel D. Salas

ICASE,
Hampton, Virginia, U.S.A.

and

Sukumar R. Chakravarthy

Metacomp Technologies Inc.,
Westlake Village, California, USA

SPRINGER-SCIENCE+BUSINESS MEDIA, B.V.

A C.I.P. Catalogue record for this book is available from the Library of Congress.

ISBN 978-94-010-6173-5 ISBN 978-94-011-5169-6 (eBook)
DOI 10.1007/978-94-011-5169-6

The cover illustration shows the ejection of water into air from a nozzle driven by a 10 microsecond impulse in the shape of 1/2 of a cosine wave. The domain is 60 x 240 microns and has been covered by a uniform square grid of 32 x 128 cells. This computation was conducted by Igor D. Aleinov and E.G. Puckett in the Department of Mathematics at the University of California, Davis using a high-order accurate projection method with a volume-of-fluid interface tracking algorithm, a Cartesian grid model of the geometry and a generalized continuum surface force model of surface tension.

Printed on acid-free paper

TABLE OF CONTENTS

CHAPTER III: DISCONTINUITIES

CHAPTER IV: OTHER APPLICATIONS

CHAPTER V: CONVERGENCE

PREFACE

This volume contains the proceedings of the ICASE/LaRC workshop, "Barriers and Challenges in Computational Fluid Dynamics," conducted by the Institute for Computer Applications in Science and Engineering and NASA Langley Research Center during August 5–7, 1996. This workshop and the proceedings are sequels to the ICASE/LaRC workshop held in August 1991 entitled, "Algorithmic Trends for Computational Fluid Dynamics (CFD) in the 90's" and the bound proceedings with the same title. Significant developments have taken place since the first workshop and this book brings together the key developments in a single unified volume.

As a new millennium approaches, Computational Fluid Dynamics (CFD) is undergoing an important transition. It is playing an integral role in many inter-disciplinary applications. In addition, CFD techniques are increasingly being used in nontraditional areas such as material sciences and manufacturing technologies, and others. New algorithmic issues arise in such contexts and need to be addressed. At this time, therefore, it is particularly important to review the challenges that face the field and identify the barriers to be overcome. In addition, techniques deemed inappropriate in CFD may well be the methods of choice when applied to new disciplines. The objectives of the workshop were two-fold:

- Identify the barriers and challenges facing CFD and propose strategies to overcome them
- Identify new application areas that can exploit CFD techniques and propose new algorithms to deal with emerging disciplines.

The present volume is comprised of five chapters, with an invited paper leading off each chapter. The first chapter "Accuracy" begins with an invited paper by M.B. Giles entitled "Progress in Applied Numerical Analysis for Computational Fluid Dynamics." This paper discusses the role of numerical analysis in helping to understand the issues of accuracy of one-dimensional shock capturing, stability of aerothermal coupling, accuracy of aeroelastic coupling and the stability of Navier-Stokes computations on unstructured grids. The rest of the chapter consists of contributions by B. Engquist and B. Sjögreen, J.W. Goodrich, M.H. Carpenter and J. Casper, and R. Lowrie et al. The second chapter "Boundary Conditions and Stiffness Issues" leads off with an invited paper by W.G. Habashi and co-workers entitled "Anisotropic Mesh Adaptation: A Step Towards a Mesh-Independent and User-Independent CFD." This paper presents the details of a mesh adaptation strategy and demonstrates convergence of the strategy for a variety of inviscid and viscous flows. This paper is followed by contributions from S.V. Tsynkov, and M. Arora and P.L. Roe. The third chapter "Discontinuities" begins with an invited paper by Osher and coworkers entitled "Numerical Methods For A One-Dimensional Interface Separating Compressible and Incompressible Flows." This paper presents a method in one dimension for tracking the interface between a droplet governed by the incompressible Navier-Stokes equations and the surrounding compressible, chemically reacting, multi-species gas also governed by the Navier-Stokes equations. This paper is followed by contributions from R.L. Löhner, G. Puckett, and D. Jacqmin. The fourth chapter "Other Applications" leads off with an invited paper by P.L. Roe entitled "Compounded of Many Simples: Reflections on the Role of Model Problems in CFD." This paper addresses two topics, the first concerning the choice of suitable model problems for a differential system, and the second topic describing a methodology that simultaneously optimizes the grid and the discrete scheme by minimizing a desired objective function. This paper is followed by contributions from R.K. Agarwal, W. Dai and P.R. Woodward, T. Linde and P.L. Roe, and N. Aslan and T. Kammash. The final chapter "Convergence" begins with a paper by B. van Leer entitled "Local Preconditioning: Who Needs It?" This paper reviews the progress made in the area of local preconditioning for the Euler and Navier-Stokes equations to equilibrate the eigenvalues and improve convergence as well as accuracy at low Mach numbers. This paper is followed by contributions from E. Turkel and P.L. Roe, and D. Sidilkover.

The editors would like to thank all the participants for their contributions and cooperation in making the workshop a success. The efforts of Ms. Emily Todd in organizing the workshop and collecting the papers, and the editorial assistance of Mrs. Shannon Verstynen are deeply appreciated.

S.R. Chakravarthy
M.D. Salas
V. Venkatakrishnan

ACCURATE AND ROBUST METHODS FOR VARIABLE DENSITY INCOMPRESSIBLE FLOWS WITH DISCONTINUITIES

W. J. RIDER AND D. B. KOTHE
Los Alamos National Laboratory
Los Alamos, NM 87545

AND

E. G. PUCKETT AND I. D. ALEINOV
University of California, Davis
Mathematics Department
Davis, CA 95616

Abstract.

We are interested in the solution of incompressible flows possessing densities that vary both discontinuously and smoothly. Smooth density variations might be caused by temperature effects, whereas abrupt variations are present at immiscible fluid interfaces. If interfaces are present, we wish to model their gross topological changes. The design of an incompressible flow algorithm that maintains solution accuracy and simulation robustness in the presence of density variations presents challenges, especially if the variations are discontinuous and topologically complex.

We discuss the construction of robust, high-fidelity fractional-step projection methods for computing such flows. We focus on algorithms for the projection, the numerical linear algebra, and interfacial physics such as surface tension. Also discussed are algorithms for multi-dimensional advection and volume tracking. Numerical examples are presented to illustrate our current capabilities.

1. Introduction

The flow of incompressible fluids with discontinuous density variations (interfaces) occurs in widespread applications. Water/air free surface flow is

V. Venkatakrishnan et al (eds.), Barriers and Challenges in Computational Fluid Dynamics, 213-230.

a classical example, e.g., a water drop falling into a pool of water. Another important example is the filling of a cast metal mold with a molten metal alloy. Yet another is the production and transport of micron-sized ink drops during inkjet printer operation. Reliable simulation of these types of flows demands a numerical model with accuracy, fidelity, and robustness.

Accuracy is defined as the quality of deviating slightly from fact. For our purposes, this definition is refined as the measured error for a given solution. There is also a distinction between order of accuracy and numerical accuracy. For reasonable grid resolution, methods with a higher order of accuracy can be accompanied by significantly larger numerical error than the lower order method. This naturally leads to our next definition.

Fidelity is defined as exact correspondence with fact. A solution that possesses fidelity is one that is physically meaningful. A method is considered to be high-fidelity when it produces solutions that are accurate relative to the computational resources (the mesh size) applied to them. For example, interface tracking mechanisms can increase solution fidelity by maintaining interface discontinuities as the interface is advected and/or undergoing topological change.

Robustness is the property of being powerfully built or sturdy. A robust method will not fail in a catastrophic manner, but rather "degrade gracefully." Robustness implies that the algorithm can be used with confidence on a difficult problem. The degree to which the degradation is graceful is subject to interpretation. A robust method should produce physically reasonable results beyond the point where accuracy is expected or achieved.

In the next several sections we will focus on the key elements in our incompressible flow solvers. In Section 2 we introduce our projection algorithms for discretizing the incompressibility constraint in a robust manner. Next, in Section 3, we discuss issues related to solving the pressure equation effectively. The methods for computing interface and flow kinematics (advection) are discussed in Section 4. Section 5 follows with an introduction and discussion of our surface tension model. Next we present a set of sample applications to amplify our arguments. Finally, in Section 7, we conclude with a summary of various outstanding numerical issues.

2. Projection Methods

Here we introduce the principal aspects of a projection method. Our basic goal with projection methods is to advance a velocity field, $\mathbf{u} = (u, v)^T$ without regard for the solenoidal nature of $\mathbf{u}$, then recover the proper solenoidal velocity field, $\mathbf{u}^d$ ($\nabla \cdot \mathbf{u}^d = 0$). The means to this end is a projection operator, $\mathcal{P}$, which projects $\mathbf{u}^d$ out of $\mathbf{u}$:

$$\mathbf{u}^d = \mathcal{P}(\mathbf{u}).$$

The velocity $\mathbf{u}$ can be decomposed into a solenoidal vector, $\mathbf{u}^d$, and a curl-free vector, expressed as the gradient of a potential, $\boldsymbol{\nabla}\varphi$. This decomposition is written

$$\rho\mathbf{u} = \rho\mathbf{u}^d + \boldsymbol{\nabla}\varphi, \tag{1}$$

or

$$\mathbf{u} = \mathbf{u}^d + \sigma\boldsymbol{\nabla}\varphi, \tag{2}$$

where $\sigma = 1/\rho$. Taking the divergence of (2) yields an elliptic equation for φ:

$$\nabla \cdot \mathbf{u} = \nabla \cdot \sigma\boldsymbol{\nabla}\varphi. \tag{3}$$

Given the solution φ to equation (3), $\mathbf{u}^d$ results from the correction,

$$\mathbf{u}^d = \mathbf{u} - \sigma\boldsymbol{\nabla}\varphi.$$

2.1. PROJECTIONS: THE BASIC IDEA

Our fractional-step algorithm consists of a predictor step, in which the solenoidal nature of the velocity field is ignored, and a corrector step, in which a projection recovers the solenoidal velocity field. In the predictor step, the time n velocity in cell (i,j), $\mathbf{u}^n_{i,j}$, is advanced with the convection-diffusion equation:

$$\begin{aligned}\mathbf{u}^{*,n+1}_{i,j} &= \mathbf{u}^n_{i,j} - \Delta t \Bigg[(\mathbf{u}\cdot\boldsymbol{\nabla}\mathbf{u})^{n+\frac{1}{2}}_{i,j} + \sigma^{n+\frac{1}{2}}_{i,j}\mathbf{G}_{i,j}\phi^{n-\frac{1}{2}} \\ &- \frac{\nu\sigma^{n+\frac{1}{2}}_{i,j}}{2} L_{i,j}\left(\mathbf{u}^n + \mathbf{u}^{*,n+1}\right) - \mathbf{f}^{n+\frac{1}{2}}_{i,j} \Bigg],\end{aligned} \tag{4}$$

where $\mathbf{G}_{i,j}$ is the discrete gradient and $L_{i,j}$ is the discrete Laplacian. This provides a nominally second-order discretization. The advection term is discretised with an unsplit high-order Godunov method (Bell *et al.*, 1989; Colella, 1990). For variable density flows, this method is described in (Bell & Marcus, 1992; Puckett *et al.*, 1997).

Several variations of the projection implied by (4) are possible. By removing the gradient of pressure from (4), φ in equation (3) is actually a pressure rather than an increment in pressure. The form of $\mathbf{u}$ in the discrete divergence on the LHS of (3) can be chosen several ways, e.g., the advanced-time predictor velocity, $\mathbf{u}^{*,n+1}$, or the predicted change, $(\mathbf{u}^{*,n+1} - \mathbf{u}^n)$. One might assume these differences to be higher order effects, but experience has shown otherwise for both exact and approximate projections, as discussed later.

An exact projection results when the Laplacian operator (L) on the RHS of equation (3) is derived from the discrete discrete divergence (D) and gradient ($\mathbf{G}$) operators: $L = D\sigma\mathbf{G}$. The discrete velocity divergence in an exact projection is zero to within the convergence tolerance of the solution to equation (3).

The exact discrete projections given above provide a good foundation in the numerical implementation of projection methods, but have some practical difficulties. These problems are discussed in (Almgren *et al.*, 1996; Lai *et al.*, 1993). The pressure/velocity decoupling can interact poorly with localized source terms (e.g., chemical reactions), leading to instabilities. Additionally, the local decoupling renders multigrid techniques cumbersome (Howell, 1993), and hampers the implementation of adaptive grid techniques (Almgren *et al.*, 1993; Howell, 1993).

To address these problems, new types of projection algorithms have been developed. In "approximate" projections, introduced in (Almgren *et al.*, 1996), the operator L is derived from a discretization of the continuous projection operator. The discrete velocity divergence in an approximate projection is not zero, but is rather a function of the truncation error. The operators D and $\mathbf{G}$ have the same form as the exact projection, but the Laplacian is modified.

2.2. ROBUST PROJECTION METHODS

In approximate projections, the velocity divergence is not constrained to be zero (to some small tolerance), hence robust algorithms can be elusive. We will demonstrate this with a single test problem, then describe improvements.

The principal problem with approximate projections is the presence and growth of null spaces in the discrete operators, which is manifested as high-frequency noise. This noise can be controlled by identifying and filtering unphysical velocity modes (Lai, 1993; Rider; 1994) or by carefully formulating the form of the approximate projection (Rider, 1994). Without these steps, approximate projection methods are prone to failure on more difficult problems.

We currently damp these spurious modes in two ways: the explicit addition of a high-order viscosity, or the use of an iterated projection derived from a discrete stencil that differs from the approximate projection stencil. These methods are most effective when used in concert.

The formulation of the projection directly affects the time evolution of the discrete divergence. If the divergence on the LHS of (3) is the difference between the predicted and old time velocity,

$$\nabla \cdot \left(\mathbf{u}^{*,n+1} - \mathbf{u}^{n}\right), \tag{5}$$

then the discrete divergence errors *accumulate* in time. On the other hand, if a predictor velocity is used,

$$\nabla \cdot \mathbf{u}^{*,n+1}, \tag{6}$$

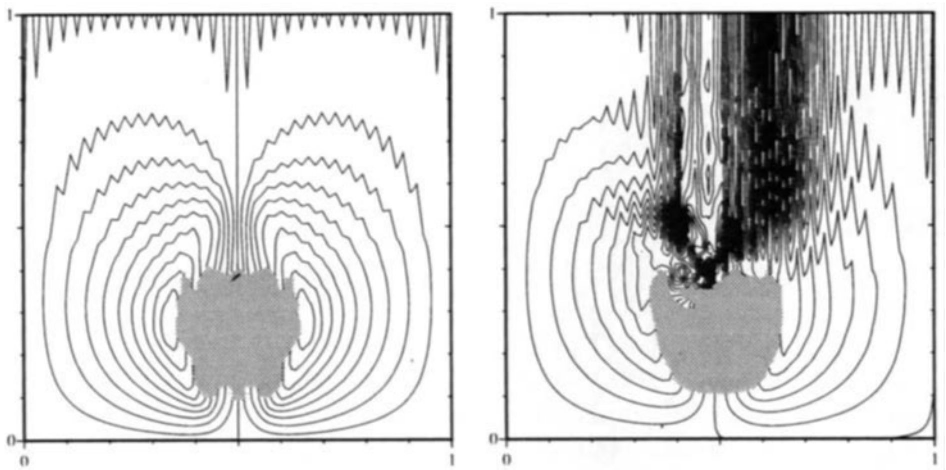
than the discrete divergence errors *do not accumulate* in time, which is preferable. It should be noted that equation (5) is the form standard in the literature.

Consider the following test problem, which will illuminate these subtle issues. A circular drop with radius 0.15 is placed at $(0.5, 0.75)$ in a unit square computational domain that is partitioned with a 64×64 grid. Gravity is unity (downward) and all boundaries are frictionless (free-slip). The drop fluid is 1000 times more dense than the background fluid (having unit density). The flow is integrated forward in time to $t = 1$ using the Euler equations. A high-order Godunov method and an unsplit piecewise linear volume tracking algorithm (discussed in Section 4) is used to advect the flow. The CFL number is $\frac{1}{2}$ unless otherwise stated. The unsteady flow is computed with variations of both the exact and approximate projection methods. Each method demonstrates second-order convergence (in space and time) on sufficiently smooth problems.

Solutions obtained with the standard exact and approximate projection methods (i.e., without filters) are shown in Figure 1[1]. Both solutions exhibit spurious features in the velocity field in the flow above the drop. The exact projection solution (Figure 1a) displays some velocity field decoupling and slight asymmetries. Despite the use of a smaller time step in integrating the flow (CFL=1/4), the approximate projection solution in Figure 1b is unacceptable. As discussed later, this solution is compromised in part because projection in equation (3) is the projecting equation (5) and solving for a pressure increment rather than a total pressure.

When the predictor velocity $(\mathbf{u}^{*,n+1})$ is projected (rather than the velocity difference) and the total pressure (rather than the pressure increment) is solved for in equation (3), the drop solutions improve significantly, as shown in Figure 2. The exact projection solution in Figure 2a now exhibits symmetry and a coupled velocity field. The approximate projection solution (Figure 2b) additionally requires velocity filters before its solution quality matches and surpasses that of the exact projection. The decoupled velocity field and asymmetries evident in Figure 1b are effectively suppressed.

[1]The standard formulation solves for an increment in pressure and projects $\nabla \cdot (\mathbf{u}^{*,n+1} - \mathbf{u})$



(a) Exact pressure increment projection of the velocity difference.

(b) Approximate pressure increment projection of the velocity difference.

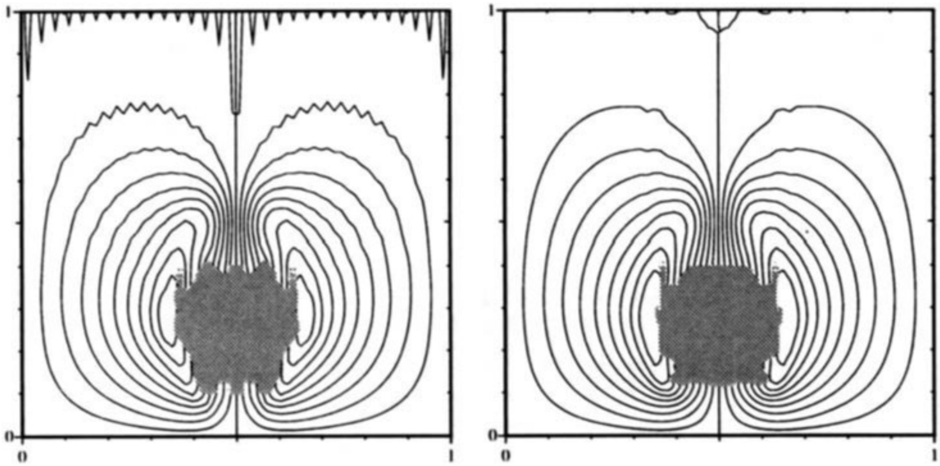
Figure 1. Drop solutions for our standard exact and approximate projections. Both solutions use a grid where all variables are cell-centered. The droplet outline and the velocity streamlines are shown. In both cases the projection equation LHS is given by equation (5).

3. Linear Algebra

The cost of incompressible flow solutions based on projection methods is typically dominated by the effort required to find solutions to the elliptic pressure equation (3). Designing an efficient and scalable method for solving this system of linear equations is therefore of paramount importance. We have used three methods in our study: a preconditioned conjugate gradient (CG) method, a multigrid (MG) method, and a multigrid preconditioned conjugate gradient (MGCG) method. Our results indicate that the MGCG method is the most effective.

3.1. CONJUGATE GRADIENT METHODS

Because the exact and approximate projections produce symmetric semi-positive-definite and positive-definite systems of linear equations, we can use CG methods (Golub & Van Loan, 1989). We can also employ preconditioning to improve convergence, which is especially important when density ratios across interfaces are large. Typically we use an incomplete Cholesky



(a) Exact pressure projection of the predictor velocity.

(b) Filtered approximate pressure projection of the predictor velocity.

Figure 2. The end product of the changes in the projection formulation and the filtering of approximate projections. The projection equation LHS is given by equation (6).

method or some other iterative procedure (SSOR, Jacobi) in finding approximate solutions to the preconditioning equation.

Several properties of preconditioned CG method's ability to solve equation (3) are worth noting. First, it is extremely robust (having never failed in our experience), and is also general, being applicable to any of our exact or approximate projections. On the other hand, the amount of work required by the preconditioned CG method grows as $N^{3/2}$ (N is the total number of unknowns). Therefore the CG method demands an ever-increasing proportion of the CPU time as the grid is refined.

3.2. MULTIGRID METHODS

Solutions to the linear systems arising from approximate projections can also be obtained with a MG algorithm (Briggs, 1987). Use of a MG algorithm is desirable because of its attractive scaling. The operation count for classical direct linear algebra solution techniques (e.g., Cholesy) scale like N^3. This scaling improves to N^2 for banded solvers that take advantage of the structure of the linear system. MG scales linearly with N. Thus, MG (where it works) will eventually provide the fastest route to a solution as the grid is refined.

One of the most important tasks in formulating a MG algorithm is the approximation of the coarse-grid linear equations. One approach is using intergrid transfer functions to define variational or Galerkin coarse grid operators. The complication and expense of this task, however, has motivated a simpler approach based on suggestions in (Liu *et al.*, 1992; Lai, 1993). The operator remains the same, as do the boundary conditions on the course grids, but σ must be defined at each level. The basic idea is to construct coarse grid approximations that produce an average cell value for $\sigma\nabla\varphi$ that is identical to the fine-level value. In using the cell-centered MG framework, this control volume derivation of the equations comes quite naturally.

We find that our MG algorithm converges quickly to a solution in most cases, but fails on occasion with flows having interfaces possessing large density variations and complex topology (e.g., a drop splashing into a pool).

3.3. MULTIGRID PRECONDITIONED CONJUGATE GRADIENT

Our current solution to the MG robustness problem is to employ the symmetric MG algorithm to precondition a standard CG method. The idea was first proposed and demonstrated by Tatebe (Tatebe, 1993). Experimentation has proven the utility of combining these two methods. This combined MGCG method usually scales like a MG algorithm, but occasionally CG-like scaling is exhibited. Nevertheless, we have found it to be robust. Ultimately we wish to design a robust method that consistently exhibits MG-like scaling.

4. Advection and Interface Tracking

4.1. HIGH-ORDER GODUNOV METHODS

We now discuss the basic principles of our advection method, which is based on the framework established in (van Leer, 1984; Bell *et al.*, 1988; Colella, 1990). Our method has many similarities with Colella's formulation, as modified for incompressible flow algorithms based on projection methods (Bell *et al.*, 1989). These methods are "unsplit", i.e., a full multi-dimensional solution is updated in a single time step. Single-step, multi-dimensional integration is important because the incompressibility constraint is inherently multi-dimensional. The flow solver should therefore reflect the intrinsic coupling of the velocity field.

Multi-dimensional advection algorithms are constructed via the time-centered approximation of the dependent variables at cell-edges. Time-centering is accomplished with the variable's full PDE (with all terms included). While the details of the methods are given in the above-stated references, important contributions in the use of these methods have recently been made. Brown and Minion discuss the nature of these solutions when resolution is not adequate (Brown & Minion, 1995), and Minion has suggested stability-enhancing improvements (Minion, 1996).

4.2. VOLUME TRACKING OF INTERFACES

The essential features of volume tracking methods are as follows. First, an initial (known) fluid interface geometry is used to generate initial fluid volume fractions in each computational cell. This task requires computing the volume truncated by the fluid interface in each cell containing an interface. Exact interface information is then discarded in favor of the discrete volume fraction data.

Interfaces are subsequently "tracked" by evolving fluid volumes in time with the solution of a standard advection equation. At any time in the solution, an exact interface location is not known, i.e., a given distribution of volume fraction data does not guarantee a unique interface topology. Interface geometry is instead inferred (based on assumptions of the particular algorithm) and its location is "reconstructed" from local volume fraction data. Interface locations are then used to compute the volume fluxes necessary for the advective term in the volume evolution equation. Volume fluxes are therefore approximated geometrically rather than algebraically. Typical implementations of these algorithms are one-dimensional, with multidimensionality built up through operator splitting. The assumed interface geometry, interface reconstruction, and volume flux calculation typically comprise the unique features of a given volume tracking method.

Our piecewise linear volume tracking algorithm, as implemented, is straightforward, simple, and extensible. This is accomplished by drawing upon the extensive literature available in the field of of computational geometry (O'Rourke, 1993). The algorithm is robust, second-order accurate in time and space, and is constructed from a set of simple geometric functions. A detailed account of our volume tracking algorithm, including comparison with other methods, is given in (Rider & Kothe, 1996). Pilliod and Puckett (Pilliod & Puckett, 1997) also review and analyze volume tracking methods, as well as introducing their own unsplit time integration scheme.

5. Surface Tension

Our current models for interfacial surface tension begin with methodology established in the continuum surface force (CSF) method (Brackbill *et al.*, 1992). The basic premise of the CSF method to model physical processes specific to and localized at fluid interfaces (e.g, surface tension) by applying the process to fluid elements everywhere within interface transition regions. Surface processes are replaced with volume processes whose integral effect properly reproduces the desired interface physics. This approach falls under the general class of immersed interface methods (Leveque & Li, 1994) whose origin dates back to the pioneering work of Peskin (Peskin, 1977). The CSF method lifts all topological restrictions without sacrificing accuracy, robustness, or reliability. It has been verified extensively in 2-D flows through its implementation in a classical algorithm for free surface flows (Kothe *et al.*, 1991; Kothe & Mjolsness, 1992), where complex interface phenomena such as breakup and coalescence have been modeled.

In the CSF model, surface tension is reformulated as a volumetric force given by

$$\mathbf{F_s} = \mathbf{f_s}\delta_s \,. \tag{7}$$

Here δ_s is a surface delta function and $\mathbf{f_s}$ is the surface tension force per unit interfacial area (Brackbill *et al.*, 1992):

$$\mathbf{f_s} = \tau\kappa\hat{\mathbf{n}} + \nabla_s\tau \,, \tag{8}$$

where τ is the surface tension coefficient, ∇_s is the surface gradient, $\hat{\mathbf{n}}$ is the interface unit normal, and κ is the mean interfacial curvature:

$$\kappa = -(\nabla\cdot\hat{\mathbf{n}}) \,. \tag{9}$$

The first term in (8) is a force acting normal to the interface, proportional to the curvature κ. The second term is a force acting along the interface (tangentially) toward regions with higher surface tension coefficient values. The normal force tends to smooth and propagate regions of high curvature, whereas the tangential force tends to force fluid along the interface toward regions of higher τ.

The surface delta function was proposed in the original CSF model to be (Brackbill *et al.*, 1992)

$$\delta_s = \frac{|\nabla c|}{[c]} = |\mathbf{n}| \tag{10}$$

where c is the characteristic (color) function uniquely identifying each fluid in the problem and $[c]$ is the jump in the color function across the interface in question, which is unity since volume fractions serve as the color function in this work. If a wide stencil is used for $\hat{\mathbf{n}}$ in (10), then δ_s will be nonzero in cells that are in close proximity to the interface. We currently force δ_s to be zero in these cells, which causes the CSF to be non-zero only within the interface transition region. A proper δ_s insures that the CSF is normalized to recover the conventional description of surface tension as the local product $\kappa h \to 0$.

Despite the success of the CSF model and related immersed interface methods, outstanding issues remain (Kothe *et al.*, 1996). If these issues can be resolved adequately, a wider range of surface tension-driven flows will be modeled reliably. For example, improved forms for δ_s, displaying better convergence and smoothness properties, are needed. Our current numerical results are very sensitive to the form used for δ_s, indicating that the quality of CSF model relies heavily on the quality of the form used for δ_s (Kothe *et al.*, 1996). Recent results by Aleinov and Puckett (Aleinov & Puckett, 1995) motivate the use of other kernels, such as the Peskin kernel (Peskin, 1977) or higher-order Nordmark (Nordmark, 1991) kernel.

Perhaps the most stringent test for a surface tension model is a test of the ability to maintain an equilibrium (minimal energy) configuration. A 2-D or 3-D static drop is such an example (Kothe *et al.*, 1996). Here a perfectly spherical drop is placed in a lighter-density background fluid, and all forces are ignored except the drop interfacial surface tension. The drop should remain stationary, as the net surface tension force is zero. An incompressible flow solution for this system, however, generates false flow dynamics (dubbed "parasitic currents") that can grow with time (sometimes unbounded) (Kothe *et al.*, 1996). The source of these currents originates in part with the surface tension model, as the computed pressure gradient at the drop interface does not exactly cancel the surface tension force. New developments in surface tension models must address this inability to maintain an equilibrium configuration.

6. Applications

6.1. A MOLD-FILLING PROBLEM

As an example of our current 3-D capabilities, consider the following sample "mold-filling" problem. A rectangular box, spanning $0 \leq x, y \leq 24$ and $0 \leq z \leq 30$, is partitioned with $24 \times 24 \times 30$ unit cubical cells. The box is initially filled with a quiescent background fluid having a density ten times less than the filling fluid. At time zero, filling fluid is injected with velocity $(0.0, 0.0, -88.6)$ through a hole in the box at $0 \leq x, y \leq 5$ and $z = 30$ (the top corner). The background fluid is allowed to escape through a vent at $19 \leq x, y \leq 24$ and $z = 30$ (the opposite top corner). Gravity is $(0.0, 0.0, -980.6)$, and both the background and filling fluid are assumed to be incompressible and inviscid. Surface tension at interfaces between the background and filling fluid is neglected. The filling dynamics (as inferred from the interface topology) are followed up to a time of 3.0.

As is evident from Figure 3, which depicts the filling fluid interface topology at four different times, this idealized calculation presents an energetic and rigorous test of our ability to model complex topology free surface flows encountered in the mold-filling process. Solutions here are obtained with a cell-centered approximate pressure projection method without filters. Equation (6) is used for the LHS of the projection. Interfaces are tracked with our piecewise-planar volume tracking method (Kothe *et al.*, 1996). Additional results of this simulation (including animations) can be found in (Kothe *et al.*, 1995).

Next we consider another difficult interfacial flow application, namely the production and transport of ink drops in an inkjet printing process.

6.2. INKJET PRINTER NOZZLE

Surface Tension The presence of the surface delta function in the expression for the continuum surface tension force is often a source for instabilities and poor convergence of projection methods. If we suppose that the surface tension coefficient τ is constant, then the singularity can be eliminated. Upon substitution of (8) and (10) into equation (7) for the surface tension force, and using the relation $\mathbf{n}|\nabla c| = \nabla c$, we can write

$$\mathbf{F_s} = \tau\kappa\nabla c = \nabla(\tau\kappa c) - \tau c\nabla\kappa \,. \tag{11}$$

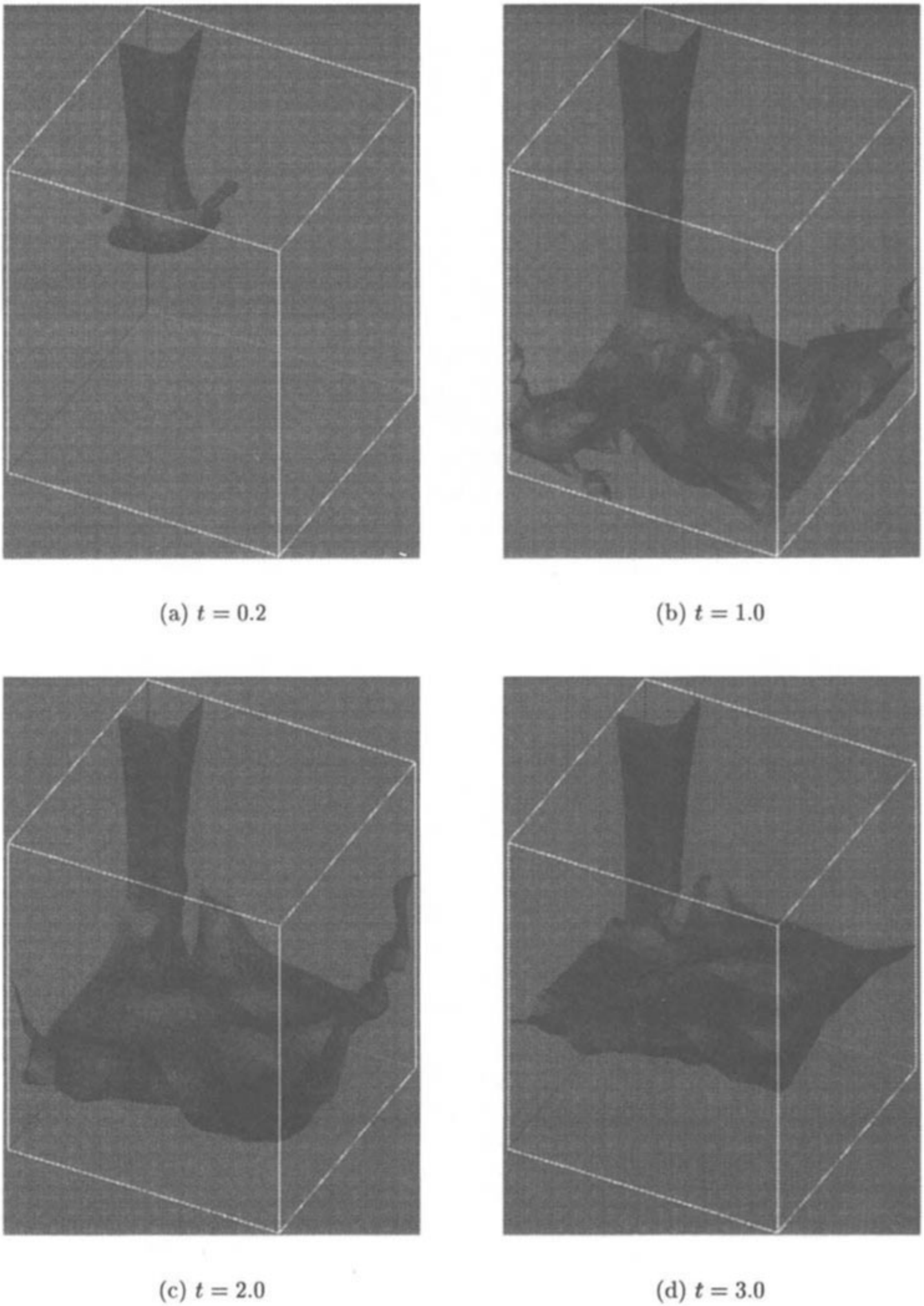
Of course, this equation is valid only if κ is defined in the entire domain and is smooth. This is not the case, since κ is defined only on the interface, but if the interface is smooth then κ can be spread smoothly over the entire domain by some simple averaging procedure. In this case the first term in the RHS of (11), which contains the main singularity, can be included into a definition of the pressure and the remaining surface tension force expression will be less singular (it will contain a Heaviside function instead of a delta function). In other words, we can introduce new definitions for the pressure and surface tension force

$$\tilde{p} = p - \sigma\kappa c \tag{12}$$

$$\tilde{\mathbf{F}}_\mathbf{s} = -\sigma c\nabla\kappa \,. \tag{13}$$

for which the Navier-Stokes equations will have the same form, except the volumetric force will not have a delta function. This method was used by Sussman (Sussman *et al.*, 1996) with a level set method. We have found this method to be more robust than the standard CSF method. It enabled us to perform computations for which the standard CSF method failed (e.g., the computation presented here).

Geometry If the computational domain has a complex geometry, special techniques are required to impose the boundary conditions. We use a Cartesian grid, in which a regular rectangular grid is cut by external boundaries. Such a grid remains rectangular away from the boundary but has irregular "incomplete" cells on the fluid-body interface. The algorithms described above need to be reformulated for the irregular boundary cells.



(a) $t = 0.2$

(b) $t = 1.0$

(c) $t = 2.0$

(d) $t = 3.0$

Figure 3. Three-dimensional simulation of a box being filled with a heavier fluid from the top corner. Shown is the filling fluid interface topology (the 0.5 volume fraction isosurface) at four different times.

The problem that arises in the velocity advection algorithm is due to the CFL constraint that requires a time step size to be proportional to the cell size. Since we want to model an arbitrary geometry, we may have very small cells on the fluid-body boundary. This places a severe restriction on the time step size, slowing down the computations and causing other difficulties. We solve this problem using the flux redistribution algorithm, described in (Almgren *et al.*, 1997), which allows us to use a time step size computed according to the regular cell CFL constraint. The general idea is to first advect velocities using a conservative transport algorithm while ignoring the fluid-body boundary. Next, in each boundary cell we apply a correction computed according to the stability requirement. Extra fluxes resulting from this correction are then distributed over the neighboring cells. Though stability of this algorithm has not been proven, its behavior is robust in our experience.

A general approach to elliptic solvers is to use a standard stencil for elliptic operator away from the boundary, but modify it for irregular cells. We use a finite element approach to the approximate projection in which pressure is located at the vertices of a regular grid. This algorithm is transferred to a Cartesian grid by changing the domain of integration in the weak form of the evolution equations. Integrals over the entire cells are replaced by integrals over those parts of the cells which are inside the fluid. The viscous solver can also be reformulated for a Cartesian grid, though it requires more work. At the moment we use a "stair-step" approximation for the viscous solver in which boundary cells are treated as if they are entirely in the fluid and no-slip boundary conditions are imposed at edges entirely in the wall.

Volume tracking methods are more complicated on Cartesian grids, since they need an interface to be reconstructed and transported within non-rectangular cells. Currently we employ a "stair-step" approximation for interface reconstruction in those cells. Another issue in the presence of multiple fluids and boundary cells are contact angles between fluid-fluid interfaces and the fluid-body boundary. At the moment we allow only two contact angles for wettable and non-wettable surfaces.

Computations Figures 4 and 5 depict computations made for ejection of water into air from the nozzle of an ink jet printer. The surface inside the nozzle is assumed to be wettable and the surface outside the nozzle to be non-wettable. Although our numerical method is purely two-dimensional, the surface tension force is computed using cylindrical symmetry. This is necessary, because curvature in the third dimension is significant in such problems, therefore neglecting it gives unrealistic results. Uniform Dirichlet boundary conditions on velocity are used on the lower boundary. Other boundary conditions for velocity are free flow on the upper boundary and

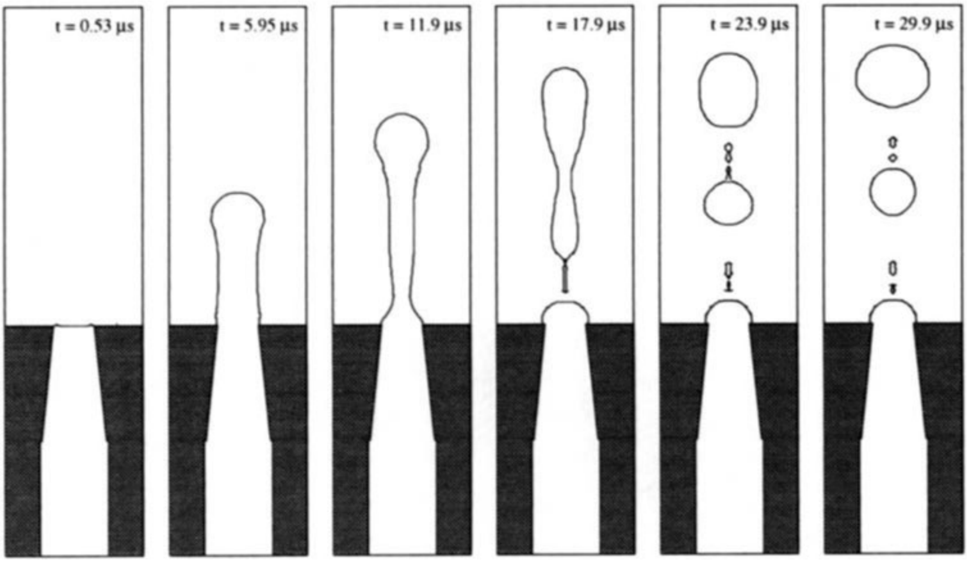


Figure 4. Ejection of water into air from a nozzle: 1/2 cosine wave impulse. The dimensions of the domain are 60 x 240 microns, with the duration of the impulse being 10 μs. The grid is 32 $\times$ 128 cells. The fluid densities are 1000 kg/m^3 and 1 kg/m^3 with viscosities of 0.001137 kg/m s and 0.00001776 kg/m s.

no-slip on the left and right boundaries. The inflow velocity at the bottom boundary is given by an impulse model using 1/2 and 1/4 of a cosine wave to $\approx$ 30 μs, followed by no further inflow. In both cases the duration of the impulse and the amount of the injected fluid are the same, except the impulse shapes are slightly different. We are interested in following the formation of the droplet and its separation from the nozzle. In both cases we observe the formation of a satellite, with its size depending on the shape of the impulse. The slightly unphysical meniscus shape at 0.53 microseconds is due to our initial conditions (flat surface) being incompatible with the specified contact angle.

7. Next Steps

We have made significant progress in achieving the goal of constructing accurate, robust solvers for variable density incompressible flows with discontinuities. Despite significant progress, much work remains. Application of our methodology to a wider variety of flows is focusing our attention on current weaknesses. Chief among these is the extension of our method to nonorthogonal, three-dimensional grids, which is currently being pursued in a new casting simulation tool (Kothe *et al.*, 1995). Additional research effort is targeting improved surface tension models, as their robustness and

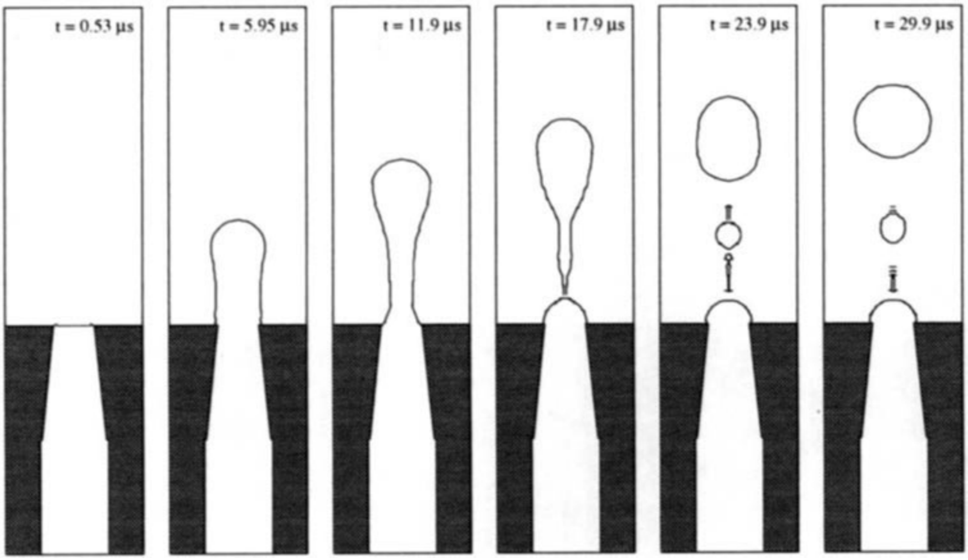


Figure 5. Ejection of water into air from a nozzle: 1/4 cosine wave impulse.

accuracy is currently less than satisfactory. Efficient linear algebra on 3-D complex grids is needed, hence will also be the focus of future efforts.

References

Aleinov, I. and Puckett, E. G. Puckett, "Computing Surface Tension with High-Order Kernels," *6th International Symposium on Computational Fluid Dyanamics*, 1995.

Almgren, A. S., Bell, J. B., Colella, P., and Howell, L. H., "An adaptive projection method for the incompressible Euler equations," *Proceedings of the AIAA Eleventh Computational Fluid Dynamics Conference*, J.L. Thomas, ed., AIAA Paper 93-3345, pp. 530-539, 1993.

Almgren, A. S., Bell, J. B., Colella, P., and Marthaler, T., "A Cartesian grid projection method for the incompressible Euler equations in complex geometries," *SIAM J. Sci. Comput.*, Vol. 18, No. 1, September 1997.

Almgren, A. S., Bell, J. B., and Szymczak, W. G., "A numerical method for the incompressible Navier-Stokes equations based on an approximate projection," *SIAM Journal of Scientific Computing*, Vol. 17, 1996, pp. 358-369.

Bell, J. B., Colella, P., and Glaz, H. M., "A second-order projection method of the incompressible Navier-Stokes equations," *Journal of Computational Physics*, Vol. 85, 1989, pp. 257-283.

Bell, J. B., Dawson, C. N., and Shubin, G. R., "An unsplit, higher order Godunov method for scalar conservation laws in multiple dimensions," *Journal of Computational Physics*, Vol. 74, 1988, pp. 1-24

Bell, J. B. and Marcus, D. L., "A second-order projection methòd variable-density flows," *Journal of Computational Physics*, Vol. 101, 1992, pp. 334-348.

Brackbill, J. U., Kothe, D. B., and Zemach, C., "A continuum method for modeling surface tension," *Journal of Computational Physics*, Vol. 100, 1992, pp. 335-354.

Briggs, W. L., "A Multigrid Tutorial," *SIAM*, 1987.

Brown, D. L. and Minion, M. L., "Perfromance of underresolved two-dimensional incompressible flow simulations," *Journal of Computational Physics*, Vol. 122, 1995, pp. 165-183.

Colella, P., "Multidimensional upwind methods for hyperbolic conservation laws," *Journal of Computational Physics*, Vol. 87, 1990, pp. 171-200.

Golub, G. H. and Van Loan, C. F., *Matrix Computations*, Johns Hopkins University Press, 1989.

Howell, L. H., "A multilevel adaptive projection method for unsteady incompressible flow," *Proceedings of the Sixth Copper Mountain Conference on Multigrid Methods*, N. D. Melson, T. A. Manteuffel, and S. F. McCormick, eds., 1993, pp. 243-257.

Kothe, D. B. et al., "Computer simulation of metal casting processes: A new approach," Technical Report LALP-95-197, Los Alamos National Laboratory, 1995. Available on World Wide Web at `http://gnarly.lanl.gov/Telluride/Telluride.html`.

Kothe, D. B. and Mjolsness, R. C., "Ripple: A new model for incompressible flows with free surfaces," *AIAA Journal*, Vol. 30, 1992, pp. 2694-2700.

Kothe, D. B., Mjolsness, R. C., and Torrey, M. D., "Ripple: A computer program for incompressible flows with free surfaces," Technical Report LA-12007-MS, Los Alamos National Laboratory, 1991.

Kothe, D. B., Rider, W. J., Mosso, S. J., Brock, J. S., and Hochstein, J. I., "Volume tracking of interfaces having surface tension in two and three dimensions," Technical Report AIAA 96-0859, AIAA, 1996. Presented at the 34rd Aerospace Sciences Meeting and Exhibit.

Lai, M. F., "A Projection Method for Reacting Flow in the Zero Mach Number Limit," Ph.D. thesis, University of California at Berkeley, 1993.

Lai, M., Bell, J. B., and Colella, P., "A projection method for combustion in the zero Mach number limit," *Proceedings of the AIAA Eleventh Computational Fluid Dynamics Conference*, J.L. Thomas, ed., 1993, pp. 776-783; AIAA Paper 93-3369.

Leveque, R. J. and Li, Z., "The immersed interface method for elliptic equations with discontinuous coefficients and singular sources," *SIAM Journal on Numerical Analysis*, Vol. 31, 1994, pp. 1019-1044.

Liu, C., Liu, Z., and McCormick, S., "An efficient multigrid scheme for elliptic equations with discontinuous coefficients," *Communications in Applied Numerical Methods*, Vol. 8, 1992, pp. 621-631.

Minion, M. L., "On the stability of Godunov projection methods for incompressible flow," *Journal of Computational Physics*, Vol. 123, 1996, pp. 435-449.

Nordmark, H. O., "Rezoning for higher order vortex methods," *Journal of Computational Physics*, Vol. 97, 1991, pp. 366-397.

O'Rourke, J., *Computational Geometry in C*, Cambridge, 1993.

Peskin, C. S., "Numerical analysis of blood flow in the heart," *Journal of Computational Physics*, Vol. 25, 1977, pp. 220-252.

Pilliod, J. E., Jr. and Puckett, E. G., "Second-Order Volume-of-Fluid Algorithms for Tracking Material Interfaces," in preparation.

Puckett, E. G., Almgren, A. S., Bell, J. B., Marcus, D. L., and Rider, W. J., "A High-Order Projection Method for Tracking Fluid Interfaces in Variable Density Incompressible Flows," *Journal of Computational Physics*, Vol. 130, 1997, pp. 269-282.

Rider, W. J., "Filtering nonsolenoidal modes in numerical solutions of incompressible flows," Technical Report LA-UR-94-3014, Los Alamos National Laboratory, 1994. Available on World Wide Web at `http://www-xdiv.lanl.gov/XHM/personnel/wjr-/Web_papers/pubs.html`.

Rider, W. J., "The robust formulation of approximate projection methods for incompressible flows," Technical Report LA-UR-94-3015, Los Alamos National Laboratory, 1994. Available on World Wide Web at `http://www-xdiv.lanl.gov/XHM/personnel-/wjr/Web_papers/pubs.html`.

Rider, W. J. and Kothe, D. B., "Reconstructing volume tracking," Technical Report LA-UR-96-2375, Los Alamos National Laboratory, 1996. Available on

World Wide Web at http://www-xdiv.lanl.gov/XHM/personnel/wjr/Web_papers-/pubs.html, also submitted to the *Journal of Computational Physics.*

Sussman, M., Almgren, A., Bell, J., Colella, P., Howell, L., and Welcome, M., "An adaptive level set approach for incompressible two-phase flows," *Proceedings of the ASME Fluids Engineering Summer Meeting*, July 1996.

Tatebe, O., "The multigrid preconditioned conjugate gradient method," *Proceedings of the Sixth Copper Mountain Conference on Multigrid Methods*, N. D. Melson, T. A. Manteuffel, and S. F. McCormick, eds., 1993, pp. 621-634.

van Leer, B., "Multidimensional explicit difference schemes for hyperbolic conservation laws," *Computing Methods in Applied Sciences and Engineering VI*, In R. Glowinski and J.-L. Lions, eds., 1984, pp. 493-497.

LIST OF ATTENDEES

Ramesh Agarwal*†
NIAR
Wichita State University
1845 Fairmount
Wichita, KS 67260-0093
(316) 978-6427
agarwal@niar.twsu.edu

W. Kyle Anderson†
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2164
w.k.anderson@larc.nasa.gov

Eyal Arian
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2208
arian@icase.edu

Necdet Aslan*†
Marmara Universitesi
Fen-Edeb, Fizik Bolumu
Ziverbey, Istanbul
TURKEY
necdet@nem.nukleer.gov.tr

Harold Atkins†
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2308
h.l.atkins@larc.nasa.gov

Abdelkader Baggag
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-9817
baggag@icase.edu

Alvin Bayliss†
Department of Engineering Sciences
Northwestern University
Technological Institute
Evanston, IL 60208
(847) 491-5585
alvin@alvin.eecs.nwu.edu

Daryl Bonhaus
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2293
d.l.bonhaus@larc.nasa.gov

*Speaker
†Panel Member

Achi Brandt†
Department of Applied Math.
The Weizmann Institute of Science
Rehovot, 76100
ISRAEL
(011) 972-8342345
mabrandt@weizmann.weizmann.ac.il

Wenlong Dai*†
University of Minnesota
116 Church Street, SE
Minneapolis, MN 55455
(612) 626-0050
wenlong@lcse.umn.edu

Mark Carpenter*†
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2318
m.h.carpenter@larc.nasa.gov

Stephen Davis
U.S. Army Research Office
P.O. Box 12211
Research Triangle Park, NC 27709-2211
(919) 549-4284
sdavis@aro.ncren.net

Sukumar Chakravarthy*†
Metacomp Technologies, Inc.
650 S. Westlake Blvd., Suite 203
Westlake Village, CA 91362-3804
(805) 371-8750
sukumarcr@aol.com

Rosa Donat
University of Valencia
Dept. de Matematica Aplicada
c/Dr. Moliner, 50
Burjassot 46100
SPAIN
donat@godella.matapl.uv.es

Chau-Lyan Chang
High Technology Corporation
28 Research Drive
Hampton, VA 23666
(757) 865-0818
chang@htc-tech.com

Michael Giles*†
Oxford University Computing Laboratory
Wolfson Building
Parks Road
Oxford OX1 3QD
UNITED KINGDOM
(011) 44-1865-273862
giles@comlab.oxford.ac.uk

John Goodrich*†
Mail Stop 5-11
NASA Lewis Research Center
Cleveland, OH 44135
(216) 433-5922
fsgdrch@jgoodrich.lerc.nasa.gov

Wagdi G. Habashi*†
Department of Mechanical Engineering
Concordia University
1455 de Maisonneuve Blvd. West, ER 301
Montreal Quebec H3GIM8
CANADA
(514) 848-3165
habashiw@cfdlab.concordia.ca

Mohamed Hafez*†
Department of Mechanical Engineering
University of California, Davis
Davis, CA 95616
(916) 752-0212
mhafez@ucdavis.edu

Ehtesham Hayder
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-4746
hayder@icase.edu

Jeffrey Hittinger
Department of Aerospace Engineering
The University of Michigan
2001 FXB Building
Ann Arbor, MI 48109-2118
(313) 764-7573
jhitt@engin.umich.edu

W. H. Hui
Department of Mathematics
The Hong Kong University of Science and Technology
Clear Water Bay
Kowloon, HONG KONG
(011) 852-2358-7415
whhui@uxmail.ust.hk

David Jacqmin*†
Mail Stop 5-11
NASA Lewis Research Center
Cleveland, OH 44122
(216) 433-5853
fsdavid@tess.lerc.nasa.gov

Leland Jameson
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2191
lmj@icase.edu

Jim Jones
Center for Applied Scientific Computing (CASC)
Lawrence Livermore National Laboratory
POB 808, L-561
Livermore, CA 94551
(510) 423-5194
jjjones@llnl.gov

Torbjorn Larsson
Prosolvia R&T, Inc.
2855 Coolidge Highway
Suite 104
Troy, MI 48084
(810) 649-1200
tobbe@prosolvia.com

Dinesh Kaushik
Department of Computer Science
Old Dominion University
Norfolk, VA 23529-0162
(757) 683-3266
kaushik@cs.odu.edu

Timur Linde*†
Department of Aerospace Engineering
University of Michigan
2004 FXB Bldg.
Ann Arbor, MI 48109
(313) 763-2397
linde@engin.umich.edu

David E. Keyes
Department of Computer Science
Old Dominion University
Norfolk, VA 23529-0162
(757) 683-4928
keyes@cs.odu.edu

Rainald Löhner*†
Institute for Computational Sciences
and Informatics
George Mason University
Mail Stop 5C3
Fairfax, VA 22030-4444
(703) 993-4075
lohner@beethoven.gmu.edu

Dennis Lankford
Sverdrup Tech., Inc.
1099 Avenue C
Arnold Air Force Base, TN 37389-9013
(615) 454-5825
lankford@hap.arnold.af.mil

Robert Lowrie*†
Los Alamos National Laboratory
CIC-19, MS B256
Los Alamos, NM 87545
(505) 667-2121
lowrie@lanl.gov

Michele Macaraeg
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2295
m.g.macaraeg@larc.nasa.gov

Robert MacCormack*†
Department of Aeronautics and
Astronautics
Stanford University
Durand Building
Stanford, CA 94305
(415) 723-4627

Laurence Manzi
Applied Physics Laboratory
Johns Hopkins University
Laurel, MD 20723-6099
(301) 953-5000
larry_manzi@jhuapl.edu

Dimitri Mavriplis
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2213
dimitri@icase.edu

Duane Melson
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2227
n.d.melson@larc.nasa.gov

Stanley Osher*†
Department of Mathematics
University of California, Los Angeles
Los Angeles, CA 90095
(310) 825-1758
sjo@math.ucla.edu

Sergei Pevchin
AOE Department
VPI & SU
P.O. Box 11705
Blacksburg, VA 24062-1705
(540) 231-6242
pevchin@aoe.vt.edu

Alex Pothen
Department of Computer Science
Old Dominion University
Norfolk, VA 23529-0162
(757) 683-4414
pothen@cs.odu.edu

Elbridge Gerry Puckett*†
Department of Mathematics
University of California, Davis
Davis, CA 95616
(916) 757-2839
puckett@math.ucdavis.edu

Philip Roe*†
Department of Aerospace Engineering
University of Michigan, Ann Arbor
Ann Arbor, MI 48109-2140
(313) 764-3394
philroe@caen.engin.umich.edu

Scott Rimbey
Department of Math Sciences
University of Delaware
Newark, DE 19711
(302) 831-1868
rimbey@math.udel.edu

Dave Rudy
Mail Stop 139
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2297
d.h.rudy@larc.nasa.gov

Thomas Roberts
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-6804
t.w.roberts@larc.nasa.gov

Yousef Saad
Department of Computer Science
University of Minnesota
4-192 Ee/CSci Building
200 Union Street
Minneapolis, MN 55455
(612) 624-7804
saad@cs.umn.edu

Kevin Roe
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-7368
kproe@icase.edu

Manuel Salas
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2174
salas@icase.edu

David Sidilkover*†
ICASE
Mail Stop 403
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-7312
sidilkov@icase.edu

R. Charles Swanson
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2235
r.c.swanson@larc.nasa.gov

Bjorn Sjogreen*†
Department of Scientific Computing
University of Uppsala
Box 120
Uppsala 751 04
SWEDEN
(011) 46-18-18-29-72
bjorns@tdb.uu.se

Shlomo Ta'asan*†
Department of Mathematics
Canegie Mellon University
Pittsburgh, PA 15213
(412) 268-5582
shlomo@andrew.cmu.edu

Jerry South
Mail Stop 285
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-3520
j.c.south@larc.nasa.gov

James Thomas
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2146
j.l.thomas@larc.nasa.gov

Chao-Ho Sung
David Taylor Model Basin
Code 5030
Bethesda, MD 20084-5000
(301) 227-1865
sung@sigma.dt.navy.mil

Eli Turkel*†
Department of Mathematics
Tel-Aviv University
Ramat-Aviv
Tel-Aviv
ISRAEL
(011) 97236408038
turkel@math.tau.ac.il

Semyon Tsynkov*†
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2150
s.v.tsynkov@larc.nasa.gov

Bram van Leer*†
Department of Aerospace Engineering
University of Michigan
Ann Arbor, MI 48109-2118
(313) 764-4305
bram@engin.umich.edu

Veer Vatsa
Mail Stop 128
NASA Langley Research Center
Hampton, VA 23681-0001
(757) 864-2236
v.n.vatsa@larc.nasa.gov

V. Venkatakrishnan
Boeing Commercial Airplane Group
POB 3707, M/S 67-LF
Seattle, WA 98124-2207
(206) 234-3124
venkat.venkatakrishnan@boeing.com

August Verhoff*†
McDonnell Douglas Corporation
MC 1067126
P.O. Box 516
St. Louis, MO 63166
(314) 233- 6343

Jeffery White
Taitech, Inc.
AMC P.O. Box 33830
Wright Patterson Air Force Base, OH 45433-0830
(513) 255-4141
whiteja@possum.appl.wpafb.af.mil

Kun Xu
Department of Mathematics
Hong Kong University of Science and Technology
Clear Water Bay, Kowloon
HONG KONG

Osman Yasar
Oak Ridge National Laboratory
P.O. Box 2008
Mail Stop 6203
Oak Ridge, TN 37831
(423) 241-5629
yasar@ccs.ornl.gov

Helen Yee
Mail Stop 202A-1
NASA Ames Research Center
Moffett Field, CA 94035
(415) 604-4769
yee@nas.nasa.gov

Hong Zhang-Sun
Department of Mathematical Sciences
Clemson University
Clemson, SC 29634
(504) 388-4966
hongsu@math.clemson.edu

Steven Zalesak
NASA Goddard Space Flight Center
Code 934
Greenbelt, MD 20771
(301) 286-8935
zalesak@gondor.gsfc.nasa.gov

ICASE/LaRC Interdisciplinary Series in Science and Engineering

1. J. Buckmaster, T.L. Jackson and A. Kumar (eds.): *Combustion in High-Speed Flows*. 1994 ISBN 0-7923-2086-X
2. M.Y. Hussaini, T.B. Gatski and T.L. Jackson (eds.): *Transition, Turbulence and Combustion*. Volume I: Transition. 1994 ISBN 0-7923-3084-6; set 0-7923-3086-2
3. M.Y. Hussaini, T.B. Gatski and T.L. Jackson (eds.): *Transition, Turbulence and Combustion*. Volume II: Turbulence and Combustion. 1994 ISBN 0-7923-3085-4; set 0-7923-3086-2
4. D.E. Keyes, A. Sameh and V. Venkatakrishnan (eds): *Parallel Numerical Algorithms*. 1997 ISBN 0-7923-4282-8
5. T.G. Campbell, R.A. Nicolaides and M.D. Salas (eds.): *Computational Electromagnetics and Its Applications*. 1997 ISBN 0-7923-4733-1
6. V. Venkatakrishnan, M.D. Salas and S.R. Chakravarthy (eds.): *Barriers and Challenges in Computational Fluid Dynamics*. 1998 ISBN 0-7923-4855-9

KLUWER ACADEMIC PUBLISHERS – DORDRECHT / BOSTON / LONDON