Journal of Computational Physics **165,** 771–772 (2000)
doi:10.1006/jcph.2000.6660, available online at http://www.idealibrary.com on IDEAL®

Abstracts of Papers to Appear

A Projection Method for Incompressible Viscous Flow on Moving Quadrilateral Grids. David P. Trebotich* and Phillip Colella.† *Department of Mechanical Engineering, University of California, Berkeley, California 94720; and †Applied Numerical Algorithms Group, Lawrence Berkeley National Laboratory, Berkeley, California 94720.

We present a second-order accurate projection method for numerical solution of the incompressible Navier–Stokes equations on moving quadrilateral grids. Our approach is a generalization of the Bell–Colella–Glaz (BCG) predictor–corrector method for incompressible flow. Irregular geometry is represented in terms of a moving, body-fitted coordinate system in cylindrical coordinates. Mapped coordinates are used to smoothly transform in both time and space the moving domain onto a logically rectangular domain which is fixed in time. In order to treat the time-dependence and inhomogeneities in the incompressibility constraint introduced by presence of deforming boundaries, we introduce a nontrivial splitting of the velocity field into vortical and potential components to eliminate the inhomogeneous terms in the constraint, and a generalization of the BCG algorithm to treat time-dependent constraints. The method is second-order accurate in space and time, has a time step constraint determined by the advective CFL condition, and requires the solution of well-behaved linear systems amenable to the use of fast iterative methods. We demonstrate the method on the specific example of viscous incompressible flow in an axisymmetric deforming tube.

Application of Vector Calculus to Numerical Simulation of Continuum Mechanics Problems. Nicholas M. Bessonov* and Dong Joo Song.† *Institute of the Problems of Mechanical Engineering, Academy of Science of Russia, Bolshoj pr. 61, St. Petersburg 199178, Russia; and †School of Mechanical Engineering, Yeungnam University, Gyongsan 712-160, Korea.

It is well known that vector–tensor notation is a compact and natural language for the mathematical formulation of continuum mechanics problems. Here we describe the application of vector techniques to numerical simulation stating with a mathematical formulation. We provide an efficient numerical scheme and implement it as a computer program. As a result the two last steps are simplified significantly (in comparison with traditional “component” form) especially for multidimensional problems with various boundary conditions in irregular geometries where nonorthogonal meshes are applied. So more attention can be focused on the physical nature of problems. Apart from the cleaner syntax, vector notation conserves the structure of traditional numerical algorithms and solves the multidimensional problems with minimum additional programming effort. Complex practical applications of this technique are described as well.

Two-Phase Flows on Interface Refined Grids Modeled with VOF, Staggered Finite Volumes, and Spline Interpolants. I. Ginzburg and G. Wittum. Interdisziplinäres Zentrum für Wissenschaftliches Rechnen (IWR), Im Neuenheimer Feld 368, 69120 Heidelberg, Germany.

A two-phase 2D model which combines the volume of fluid method with implicit staggered finite volumes discretization of the Navier–Stokes equation is presented. Staggered finite volumes are developed on the base of nonconforming Crouzeix–Raviart finite elements, where all components of the velocity lie in the middle of the element edges and the pressure degrees of freedom are found in the centers of mass of the elements. Staggered finite volumes extend MAC regular staggered grids to unstructured meshes. A linear saddle point problem, resulting

0021-9991/00 $35.00

either from the discretization or the Newton method, is solved for all unknown pressures and velocities. The interface is represented with spline interpolants which follow the VOF distribution. Adaptive mesh refinement is used to obtain a high level of the uniform refining at the domain of dependence of the interface. The aligned grid is obtained by the irregular refining of the cells which are intersected by a curve. The boundaries of its elements coincide with the slope segments going through the intersections of the curve with the underlying regular elements boundary. The deformable computational grids are used only to discretize the Navier–Stokes equation. The advection of volume fractions is done on the advection mesh, which corresponds to highest regular refining on the computational grid. Approximation of the surface tension on spline interpolants offers a straightforward way to describe correctly the pressure jumps on interface fitted staggered grids. This allows us to delete the anomalous currents around a statical bubble and to reduce them effectively in real simulations. On the aligned grid, the continuity of the viscous stress is modeled exactly due to finite volume approach. Using the proposed numerical techniques, single bubble rise is analyzed.

An Extended FDTD Scheme for the Wave Equation: Application to Multiscale Electromagnetic Simulation. Jean-Luc Vay. Lawrence Berkeley National Laboratory.

An algorithm for the application of the mesh refinement technique to finite-difference calculation of the wave equation is presented via the introduction of a new “extended” FDTD scheme. This scheme can be viewed as an extension of the Yee scheme using a new set of variables relating to the direction of propagation of the waves along an axis. Because of the additional information it contains, this scheme allows a more natural implementation of the mesh refinement technique. The extended scheme is presented for both one-dimensional and multidimensional systems. The mesh refinement algorithm is given in one dimension and the performances are compared to those of other proposed schemes.

A Hybrid Algorithm for the Joint PDF Equation of Turbulent Reactive Flows. P. Jenny, S. B. Pope, M. Muradoglu, and D. A. Caughey. Sibley School of Mechanical and Aerospace Engineering, Cornell University, Ithaca, New York 14853.

In this paper a new particle–finite-volume hybrid algorithm for the joint velocity–frequency-composition PDF method for turbulent reactive flows is presented. This method is a combination of a finite-volume scheme and a particle method. The finite-volume scheme is used to solve the Reynolds averaged Navier–Stokes equations and the particle method to solve the joint PDF transport equation. The motivation is to reduce the bias and the statistical error and to have an algorithm which is more efficient than standalone particle-mesh methods. Therefore, in the particle method we use the smoother mean density $\langle \rho \rangle$ and Favre averaged velocity $\tilde{\mathbf{U}}$ fields computed by the finite-volume scheme: This scheme is an Euler solver for compressible flow with the turbulent fluxes and the reaction term, which are computed by the particle algorithm, as source terms. Since some of the quantities are computed twice (i.e., the mean density $\langle \rho \rangle$ and the Favre averaged sensible internal energy $\tilde{e}_s$), by the finite-volume scheme and by the particle method, the hybrid algorithm is redundant. Although the model differential equations are consistent, it was difficult to satisfy consistency numerically and an accurate particle tracking algorithm is crucial. Therefore a new sheme to interpolate the Favre averaged velocity has been developed which is second-order accurate and quasi-conservative; i.e., it is based on the fluxes at the volume interfaces. Another important issue is the coupling between the finite-volume scheme and the particle method. A new time averaging technique adds stability to the hybrid algorithm, and it also reduces the bias and the statistical error enormously. The properties of the new algorithm are demonstrated by results for a non-premixed piloted-jet flame test case. First it is shown that the solution becomes statistically stationary and that it is internally consistent. Studies of the asymptotic behavior show that, for a given error tolerance, the new hybrid algorithm requires much less computer time than the stand-alone particle-mesh method (for this piloted-jet flame test case a factor of 20 times less). Finally, grid convergence studies verify that the scheme is second-order accurate in space.