N94-23056

COMPUTATION OF PROPULSION-RELATED FLOWFIELDS USING UNSTRUCTURED ADAPTIVE MESHES

Jonathan M. Weiss and Jayathi Y. Murthy
Fluent, Inc.
10 Cavendish Ct.
Lebanon, NH 03766

Overview

In this paper, we describe a computational fluid dynamics (CFD) technique based on unstructured triangular/tetrahedral meshes. A finite-volume scheme is used in conjunction with a multi-stage Runge-Kutta algorithm. Convergence enhancements in the form of dual time-stepping and time-derivative preconditioning are used to overcome the limitations of conventional multi-stage schemes. The method is applied to propulsion-related flows and shown to perform satisfactorily.

Technical Discussion

Governing Equations and Discretization

The equations governing the unsteady flow of a multi-species fluid mixture may be written in integral Cartesian form for an arbitrary control volume, V, with surface, S, as follows:

$$\frac{\partial}{\partial t}\int\!\!\int\!\!\int \boldsymbol{W}\, dV + \int\!\!\int \boldsymbol{F}\cdot d\mathrm{S} + \int\!\!\int \boldsymbol{G}\cdot d\mathrm{S} = \int\!\!\int\!\!\int \boldsymbol{H}\, dV \tag{1}$$

where

$$\begin{aligned}
\boldsymbol{W} &= [\rho,\ \rho v_x,\ \rho v_y,\ \rho v_z,\ \rho e,\ \rho Y_l]^T \\
\boldsymbol{F} &= \left[\rho \boldsymbol{v},\ \rho v_x \boldsymbol{v} + p\hat{\boldsymbol{i}},\ \rho v_y \boldsymbol{v} + p\hat{\boldsymbol{j}},\ \rho v_z \boldsymbol{v} + p\hat{\boldsymbol{k}},\ \rho e\boldsymbol{v} + p\boldsymbol{v},\ \rho \boldsymbol{v} Y_l\right]^T \\
\boldsymbol{G} &= [0,\ \boldsymbol{\tau}_{xi},\ \boldsymbol{\tau}_{yi},\ \boldsymbol{\tau}_{zi},\ \boldsymbol{\tau}_{ij} v_j + \boldsymbol{q},\ \rho \hat{\boldsymbol{v}}_l Y_l]^T \\
\boldsymbol{H} &= [0,\ 0,\ 0,\ 0,\ \dot{\omega}_l]^T
\end{aligned}$$

Here ρ, $\boldsymbol{v}$, e, and p are the density, velocity, total energy, and pressure of the fluid mixture, respectively. Y_l and $\dot{\omega}_l$ are the mass fraction and rate of production of the l-th chemical species. $\boldsymbol{\tau}$, $\boldsymbol{q}$ and $\hat{\boldsymbol{v}}_l$ are the viscous stress tensor, and the heat flux and diffusion velocity vectors, respectively.

The domain is divided into triangular/tetrahedral volumes and the governing equations discretized over these volumes using a cell-centered finite-volume approach.[5] Odd-even decoupling due to the use of central-difference operators is damped by adding fourth difference artificial dissipation;[2] a first-order dissipative term is selectively added near discontinuities.

Preconditioning for Convergence Enhancement

The performance of the time-marching scheme at low Mach numbers is enhanced by using time-derivative preconditioning[1, 4, 7] in the context of unstructured meshes. Equation (1) is modified as follows:

$$\boldsymbol{\Gamma}\frac{\partial}{\partial t}\int\!\!\int\!\!\int \hat{\boldsymbol{W}}\, dV + \int\!\!\int \boldsymbol{F}\cdot d\mathrm{S} + \int\!\!\int \boldsymbol{G}\cdot d\mathrm{S} = \int\!\!\int\!\!\int \boldsymbol{H}\, dV$$

where $\hat{W}$ are a new set of primary dependent variables: $\hat{W} = [p, v_x, v_y, v_z, T, Y_l]^T$, and Γ is the preconditioning matrix:

$$\Gamma = \begin{bmatrix} \frac{1}{U_r^2} & 0 & 0 & 0 & 0 & 0 \\ \frac{v_x}{U_r^2} & \rho & 0 & 0 & 0 & 0 \\ \frac{v_y}{U_r^2} & 0 & \rho & 0 & 0 & 0 \\ \frac{v_z}{U_r^2} & 0 & 0 & \rho & 0 & 0 \\ \frac{h+\frac{v^2}{2}}{U_r^2} - 1 & \rho v_x & \rho v_y & \rho v_z & \rho Cp & \rho(h_l - h_N) \\ \frac{Y_l}{U_r^2} & 0 & 0 & 0 & 0 & \rho \end{bmatrix}$$

Here U_r is a reference velocity, $U_r < Min(|v|, U_c)$, where U_c is a characteristic speed of the flow such as the local speed of sound or a maximum velocity within the domain.

In the preconditioned system, time-step definition is based on the modified eigenvalues: $\lambda_\pm$, $|v|$, $|v|$, ... where

$$\begin{aligned} \lambda_\pm &= \frac{1}{2}\left[|v|\left(1 + \kappa\beta U_r^2\right) \pm c'\right] \\ c' &= \sqrt{|v|^2 (1 - \kappa\beta U_r^2)^2 + 4U_r^2} \\ \beta &= \left(\left.\frac{\partial\rho}{\partial p}\right|_T + \frac{\left.\frac{\partial\rho}{\partial T}\right|_p}{\rho C_p} \right) \end{aligned}$$

The parameter κ is included to provide time-step control at low Reynolds numbers.[1] When viscous effects become important, the preconditioning scheme alters the acoustic speed such that the CFL number is of the same order of magnitude as the von-Neumann number; thus optimal wave propagation speeds as well as optimum von-Neumann numbers result.[7]

Dual Time-Stepping for Unsteady Flows

To provide for efficient, time accurate solution of the governing equations, we employ dual time-stepping,[3] adapted for use with an explicit multi-stage scheme. Here we introduce a preconditioned pseudo-time derivative term into (1) as follows:

$$\Gamma \frac{\partial}{\partial \tau} \int\int\int \hat{W}\, dV + \frac{\partial}{\partial t} \int\int\int W\, dV + \int\int F \cdot dS + \int\int G \cdot dS = \int\int\int H\, dV \qquad (2)$$

Note that as $\tau \to \infty$, the first term on the LHS of (2) vanishes and (1) is recovered.

The time-dependent term in (2) is discretized in an implicit fashion by means of a second order accurate, three-point backwards difference in time and the pseudo-time derivative is driven to zero by means of the following multi-stage algorithm:

$$\begin{aligned} \hat{W}^{(0)} &= \hat{W}^{(k)} \\ \left[\Gamma + \frac{3}{2}\frac{\Delta\tau}{\Delta t}\frac{\partial \tilde{W}}{\partial \hat{W}}\right](\hat{W}^{(i)} - \hat{W}^{(0)}) &= -\alpha_i \Delta\tau \left\{ \mathbf{R}\tilde{W}^{(i-1)} + \frac{1}{2\Delta t}(3\tilde{W}^{(i-1)} - 4\tilde{W}^{(n)} + \tilde{W}^{(n-1)}) \right\} \\ \hat{W}^{(k+1)} &= \hat{W}^{(m)} \end{aligned} \qquad (3)$$

Here $i = 1, 2, ..., m$ is the stage counter for the m-stage scheme, and k and n represent any given pseudo-time and physical-time level, respectively. Throughout the iterations on k, $\tilde{W}^{(n)}$ and

$\tilde{W}^{(n-1)}$ are held constant. As $\tau \to \infty$, $\tilde{W}^{(k+1)} \to \tilde{W}^{(n+1)}$. Note that the matrix on the LHS of (3) is inverted in a point-wise fashion and its inverse is readily derived analytically and need not be computed numerically. Note also that the physical time step, Δt, is limited only by the level of desired temporal accuracy. And the pseudo time-step, $\Delta\tau$, is determined by the multi-stage scheme, which here employs local time stepping and preconditioning for convergence enhancement.

Results

Rocket Engine Flowfield

To demonstrate the viability of using unstructured solution-adaptive meshes to compute transonic, internal, viscous flows typically found in space propulsion applications, the finite-volume, multi-stage method presented here is used to compute a transonic, converging/diverging nozzle flowfield. Figure 1 shows the unstructured triangular mesh used in these computations. This mesh consists of 6,877 cells. It has been adapted to gradients of velocity in order to resolve the boundary layer along the nozzle walls, and to gradients of pressure in order to better capture the oblique shock in the divergent section. Pressure contours within the nozzle, plotted on a logarithmic scale in Figure 2, depict the oblique shock that develops as the expanding flow is turned inward by the nozzle walls.

Flow Past Circular Cylinder

The time accurate, dual time-stepping scheme described above is applied to solve the unsteady, two-dimensional flow over a circular cylinder in crossflow at a Reynolds number, $R_e = Ud/\nu = 75$. At this Reynolds number the flow is laminar and exhibits periodic unsteady behavior as vortices shed from the cylinder to form a Karman vortex street in the wake. The computational domain for this problem is chosen to extend 5 diameters upstream and 20 diameters downstream of the cylinder, with symmetry boundaries placed 5 diameters above and below. Details of the unstructured mesh in the vicinity of the cylinder are shown in Figure 3(a). The specified fluid is air at standard conditions with $U = 11.5\, m/s\, (M = 3x10^{-4})$ and $d = 1$ cm. The predicted shedding frequency, f, is $1.67s^{-1}$, resulting in a Strouhal number, $S_t = fd/U = 0.146$. This compares reasonably with the measured value[6] of 0.147. Contours of stream function in the vicinity of the cylinder at several times during the shedding cycle are show in Figure 3(b). The physical time step, Δt is chosen to be $0.025s$ and time periodic behavior is achieved in roughly $5s$ beginning from an impulsive start from rest at $t = 0$. Twenty-five inner iterations are performed at each time level to achieve three orders of magnitude decrease in the x-momentum residual using the preconditioned, multi-stage scheme with a CFL of 2.7. Thus 600 iterations are required to resolve one time period of $0.6s$. This represents a 1000 time speed-up over global time stepping which, for this grid and these flow conditions, would have restricted the time step in each cell to roughly $1x10^{-6}s$.

Conclusions

A finite-volume, multi-stage algorithm based on an unstructured grid topology is presented. The use of solution adaption is demonstrated in the calculation of a typical space propulsion application. Dual time-stepping and time-derivative preconditioning are shown to provide efficient solution of unsteady, low speed flow on an unstructured mesh. The benefits of extending this method to low Mach number flows with heat release, and to incompressible flows with variable density are evident, and our efforts continue in that direction.

References

[1] Choi, Y.-H., and Merkle, C. L., "Time-Derivative Preconditioning for Viscous Flows," AIAA Paper 91-1652, June 1991.

[2] Jameson, A., Schmidt, W. and Turkel, E., "Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time Stepping Schemes," AIAA Paper 81-1259, June 1981.

[3] Merkle, C.L. and Athavale, M.M., "Time-Accurate Unsteady Incompressible Flow Algorithms Based on Artificial Compressibility," AIAA Paper 87-1137.

[4] Shuen, J.S., Chen, K.H. and Choi Y.H., "A Time-Accurate Algorithm for Chemical Non-Equilibrium Viscous Flows at All Speeds," AIAA Paper 92-3639, July 1992.

[5] Smith, W., "Multigrid Solution of Transonic Flows on Unstructured Grids," *Recent Advances in Computational Fluid Dynamics, ASME Fluids Engineering Division*, FED Vol. 103, ASME, NY, 1990.

[6] Tritton, D., "Experiments on the flow past a circular cylinder at low Reynolds numbers," *J.Fluid.Mech.*, Vol. 6, pp. 547-657.

[7] Venkateswaran, S., Weiss, J.M. and Merkle, C.L., "Propulsion Related Flowfields Using the Preconditioned Navier-Stokes Equations," AIAA Paper 92-3437, July 1992.

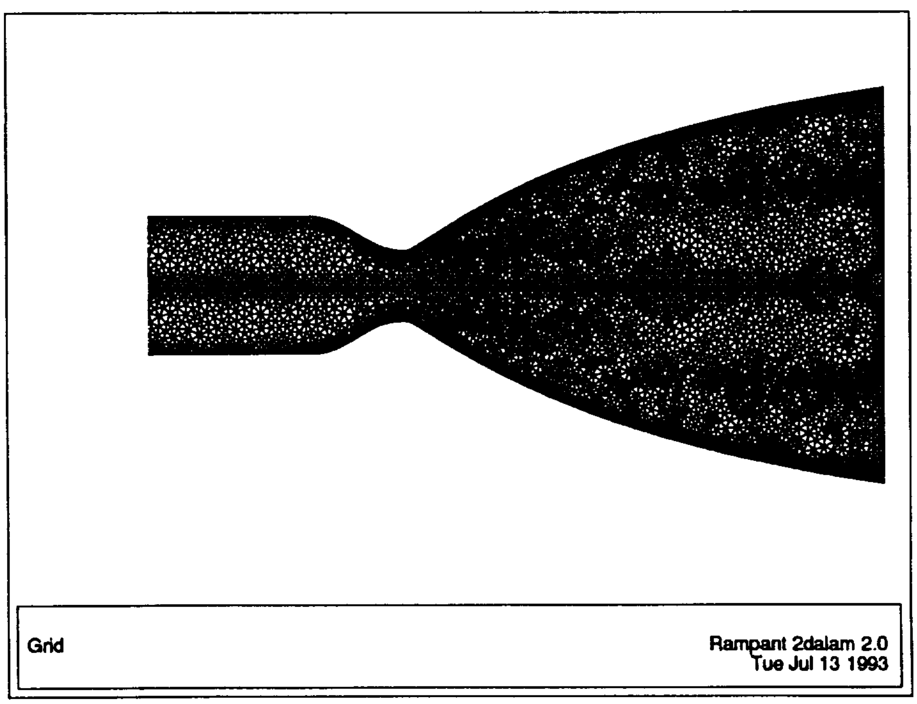

Figure 1: Unstructured, triangular mesh used for transonic, converging/diverging nozzle calculations.

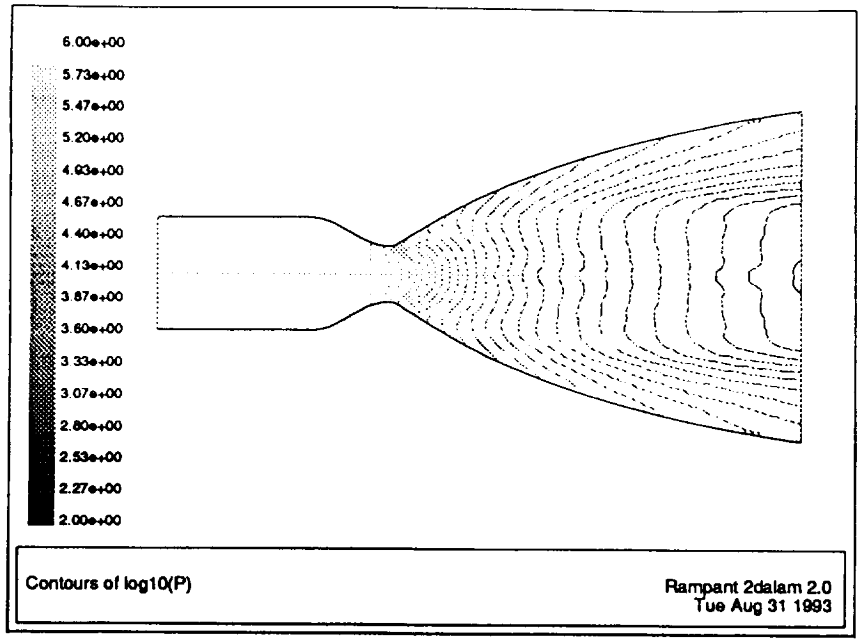

Figure 2: Pressure contours in convergent/divergent nozzle (plotted on logarithmic scale) computed using unstructured, triangular mesh.

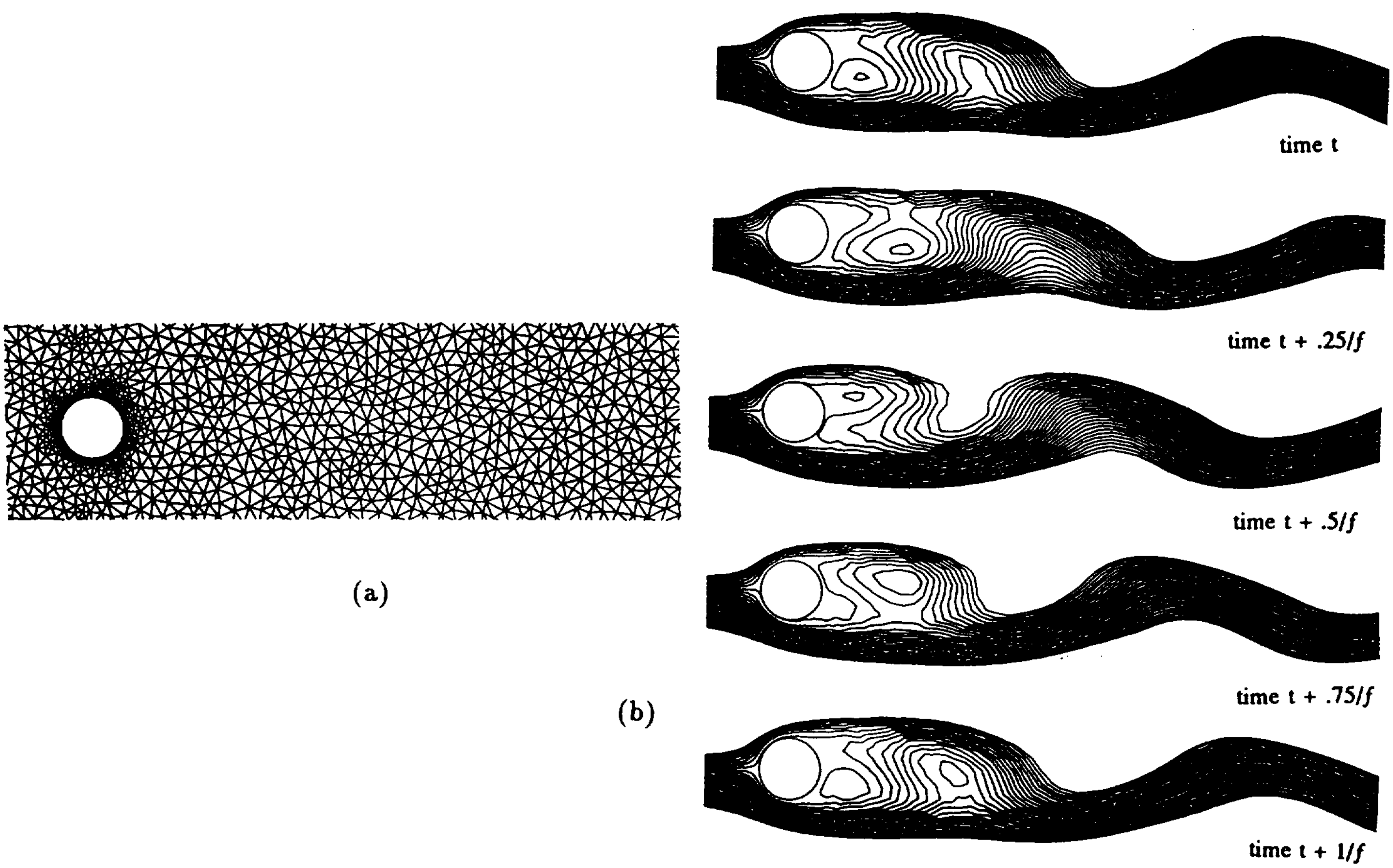

Figure 3: (a) Detail of unstructured mesh, and (b) contours of stream function about circular cylinder.