NASA Contractor Report 177630

Progress in Computational Unsteady Aerodynamics

Shigeru Obayashi

MCAT Institute
3933 Blue Gum Drive
San Jose, CA 95127

Prepared for
Ames Research Center
CONTRACT NCC2-605
November 1993

NASA
National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035-1000

PROGRESS IN COMPUTATIONAL UNSTEADY AERODYNAMICS

Shigeru OBAYASHI

MCAT Institute
NASA Ames Research Center, Mail Stop 258-1,
Moffett Field, California, 94035-1000, U.S.A.

Introduction

After vigorous development for over twenty years, Computational Fluid Dynamics (CFD) in the field of aerospace engineering has arrived at a turning point toward maturity. Many algorithms have been developed and the feasibility of CFD has been demonstrated. Now, after so many demonstrations, the next question is how to use CFD for realistic applications.

Aerospace CFD is now being asked to catch up with the sophistication of wind tunnels, which are nearly a century old for aerodynamic research.[1,2] Such expectation is probably motivated by the accelerated computerization of our work environment. We use computers to manipulate equations, calculate numbers, plot figures and write reports. CFD programs will be just one more of software packages like word processor and spreadsheet programs. Then, we won't study CFD, but we will simply use it. As computers become more affordable, CFD is expected to be a better scientific and engineering tool.

As computer hardware has become more advanced, CFD researchers have explored more complicated, computer intensive applications. The 1980's can be categorized with the compressible Reynolds averaged Navier-Stokes equations and three dimensional steady flow simulations, and the 90's as the era for unsteady flow simulations.[3]

Among various unsteady phenomena, unsteady transonic phenomena are of great interest because they include many important problems in unsteady aerodynamics,[4] such as flutter, limit cycle oscillations, maneuver aerodynamics, control reversal, buzz, gust response, active controls, unsteady shock-vortex interaction. Furthermore, local regions of unsteady transonic flows are found over a wide range of freestream speeds, for example, on the noses of bluff re-entry bodies at hypersonic speeds, on bluff bodies at low subsonic speeds, and on many wings near maximum lift at low speeds due to the high local suctions in the leading-edge region.[4] Theoretical analysis of such phenomena is complicated by the presence of mixed flow, embedded shocks, separation and vortical flow. To simulate such flow fields, computations based on the unsteady Navier-Stokes equations are needed.

This paper discusses issues related to algorithm development for the Euler/Navier-Stokes equations, code validation[5] and recent applications of CFD for unsteady aerodynamics. Algorithm development is a fundamental element for a good CFD program. Code validation tries to bridge the reliability gap between CFD and experiment. Many of the recent applications also take a multidisciplinary approach,[6] which is a future trend for CFD applications.

Algorithm Development

Structured vs. Unstructured Grids

Use of structured grids has been a driving force for CFD development (for example, see Ref. 7). Zoning and topological constraint of structured grids are often tedious for complex geometry in three dimensions, but numerical generation of structured grids for individual zones is relatively easy. Corresponding flow solvers can be highly efficient by taking advantage of structured grids. In addition, structured grids are indispensable for the high-aspect-ratio grids required for viscous flow calculations.

The zonal approach has been widely used in CFD applications because of its efficiency and versatility. However, basic questions remain in numerical techniques for handling the zonal interface. Most of the zonal methods use nonconservative interpolation at the zonal boundary.[8] Questions about how much can conservation be relaxed at the interface, time accuracy, viscous effects, turbulence model, etc. still remain to be answered. Errors are often negligible but they should be addressed quantitatively.

Since the late 80's, unstructured grid methods have attracted attention (for example, see Ref. 9-11). However, the methodology has not become fully productive yet, especially in three dimensions. The methodology consists of two parts: the unstructured grid generation and the unstructured flow solver. Complete unstructured grid generation is sometimes overwhelming and not always necessary. One can construct a mostly structured grid with a locally unstructured grid more easily. Or one can adapt a hybrid approach such as the FDM-FEM (finite-difference method and finite-element method) approach[12] and the prismatic grid approach.[13] Therefore, the majority of CFD work will remain with the structured grid approach. On the other hand, the unstructured-grid flow solver has more flexibility than the structured-grid flow solver. Thus the choice between the unstructured- and structured-grid flow solvers will be the choice between flexibility and efficiency on available computer architecture.

Freestream Capturing on Moving Grid

Freestream capturing in the discretized equation is a fundamental requirement of CFD.[14] The geometric conservation law (GCL) is known to be important for the moving grid case.[15] However, what is not always known is that it is only a necessary condition, not a sufficient one.[16] For example, one cannot use the GCL condition to compute time metrics.

Let's start from the integral form of the conservation law for a given cell:

$$\int_{t_1}^{t_2} \int Q dV dt + \int_{t_1}^{t_2} \oint n \cdot (F - vQ) dS dt = 0, \tag{1}$$

where Q is the vector of conserved quantities, F is the flux and v is the velocity of the surface element. For the freestream with Q_∞ and F_∞, one obtains

$$Q_\infty \left[V(t_2) - V(t_1) \right] + F_\infty \int_{t_1}^{t_2} \oint n dS dt - Q_\infty \int_{t_1}^{t_2} \oint n \cdot v dS dt = 0. \tag{2}$$

To satisfy Eq. (2) for any Q_∞ and F_∞, we have

$$\oint n dS = 0, \tag{3}$$

and

$$V(t_2) - V(t_1) = \int_{t_1}^{t_2} \oint n \cdot v dS dt. \tag{4}$$

Equation (3) is the mathematical representation of a closed cell, which is a requirement for any grid system to satisfy numerical conservation. The right-hand side of Eq. (4) represents a sum of the volume swept by each cell surface between the time t_1 and t_2. Thus, the equation indicates that the sum is equal to the change of the total volume. Both Eqs. (3) and (4) can be satisfied numerically by applying standard formula for computing surfaces and volumes.

Now let's look at the time differential form of Eq. (1):

$$\frac{d}{dt} \int Q dV + \oint n \cdot (F - vQ) dS = 0. \tag{5}$$

For any Q_∞ and F_∞, we have Eq. (3) and

$$\frac{dV}{dt} = \oint n \cdot v dS. \tag{6}$$

The straightforward discretization of Eq. (6) evaluates the right-hand side only at time level t_1, but the resulting discretized equation is not necessarily valid. To preserve the freestream, we have to go back to the discretized form of Eq. (4).

When we start from the differential form of Eq. (1) in both space and time, the differential forms of Eqs. (3) and (4) need to be satisfied. The discretized, differential form of Eq. (4) results in the so-called GCL condition. However, neither of the finite-difference approximations of those equations will be valid. To satisfy those equations, we have to go back to Eqs. (3) and (4) again and construct freestream capturing metrics. In three dimensions, one grid point in the finite-difference grid can be regarded as surrounded by eight finite-volume cells. Finite-difference metric terms can be derived from finite-volume geometric quantities over those eight cells.

Upwind vs. Central-Difference Schemes

Many CFD applications have used the central-difference (CD) scheme. Since the CD scheme is unstable, it is always used with artificial dissipation. The original dissipation model was the fourth-order dissipation model with a scalar coefficient.[17] Then the coefficient was replaced with the spectral radius of the flux Jacobian.[18] To mimic the TVD (total variation diminishing) method,[19] the combination of the second and fouth-order dissipation was also introduced. The latest development in the CD scheme is the matrix dissipation, which further mimics the upwind scheme by accounting all eigenvalues associated with the flux Jacobian.[20]

During the 80's, upwind schemes were developed, studied in various aspects, and widely accepted along with the TVD scheme.[21] The turning point of the upwind study was Ref. 22. Originally, the upwind algorithms were studied to obtain a better shock wave profile. However, Ref. 22 reported that the Roe upwind scheme[23] is also good at capturing a boundary layer profile. Further studies[20,24] revealed that the Roe upwind scheme shows good grid convergence for the boundary layer profile and the vortical flow field. On the other hand, the CD scheme with the spectral-radius scalar dissipation shows poor grid convergence. Since the operation count of the Roe scheme was comparable to that of the CD scheme with the matrix dissipation, the Roe scheme became very popular.

Behind its success, several failures of the Roe scheme were found in the late 80's.[14,25-27] Limitations of the linearized Riemann solver emerged and led to the development of the HLLE (Harten-Lax-van Leer-Einfeldt) scheme.[28] Currently, Wada's modified HLLE scheme is the best derivative of the HLLE scheme.[29] Also there was a renewed interest in the flux vector splitting schemes, which led to the AUSM (advection upstream splitting method)[30] and the state vector splitting schemes.[31,32]

Extension of those upwind algorithms to multi-dimensions was performed by using dimensional splitting. The inadequacy of such an extension is revealed when the grid is not aligned to the typical flow features, such as shock and shear waves. Since the late 80's, several researchers have attempted to develop robust schemes to replace the dimensional splitting upwind schemes. Interested readers may refer to, for example, Refs. 31-33 along with a critical paper, Ref. 34.

Another important issue to be addressed is vortex capturing.[35,36] It is a common experience that a computed vortex quickly dissipates as soon as the vortex separates from a solid surface. A successful vortex capturing scheme will be of great interest.

Second-Order vs. Higher-Order Schemes

There is always interest in higher-order schemes because the use of such schemes can increase computational efficiency. However, researchers often concentrate on higher-order interpolation techniques or elaborate limiter functions and miss the whole picture of the scheme.

Let's consider the space discretized form of the two-dimensional Euler equations in the curvilinear coordinates as

$$\tilde{Q}_t + \frac{\hat{E}_{i+1/2,j} - \hat{E}_{i-1/2,j}}{\Delta\xi} + \frac{\hat{F}_{i,j+1/2} - \hat{F}_{i,j-1/2}}{\Delta\eta} = 0, \tag{7}$$

where $\hat{E}_{i+1/2,j} = \hat{E}\left(Q_L, Q_R, \xi_{i+1/2,j}\right)$ is the numerical flux and the subscripts L and R indicate the left and right states at the point $(i+1/2, j)$. Linear interpolation of Q_L and Q_R leads to a second-order-accurate flux. However, higher-order representation of Q_L and Q_R do not necessarily lead to higher-order accuracy. Even if we have exact values of Q_L and Q_R, the flux difference itself remains second-order accurate:

$$\tilde{E}_\xi = \frac{\hat{E}_{i+1/2,j} - \hat{E}_{i-1/2,j}}{\Delta\xi} + O(\Delta\xi^2). \tag{8}$$

Thus, higher-order numerical fluxes must be used in higher-order representations of the spatial derivatives to obtain higher-order accurate evaluations of these derivatives.[37] In the method based on the integral form of the governing equation, they should be used in higher-order integrals of the fluxes along the boundaries of each computational cell.[37] In either way, the metric terms must be evaluated in the higher-order manner as well. When we change the representations of the metric terms, we have to go back to the discretized forms of Eqs. (3) and (4) to check the freestream capturing. Higher-order representations of the metric terms do not necessarily satisfy those equations. In addition, we have to consider the proper boundary conditions because of the enlarged stencil of grid points to obtain higher-order accuracy. All-in-all, extension beyond the second-order accuracy requires much additional work and care.

Implicit vs. Explicit Schemes

Diagonal Beam-Warming,[38,39] LU-ADI (lower-upper factored alternating direction implicit)[40,41] and LU-SGS (lower-upper factored symmetric Gauss-Seidel)[42,43] methods are the popular implicit methods. The computational effort necessary for such implicit inversions is actually less than that necessary for evaluating the explicit terms because of the costly dissipation or upwind formulation. This means that they are faster than the multistage explicit schemes.[18] Also a time step size can be much larger for the implicit scheme than one for the explicit scheme. Thus, most of the practical unsteady applications have been carried out using implicit methods.

The major drawback of those schemes is that they are limited to first-order accuracy in time. Use of a small time step usually gives satisfactory results.[41] Otherwise, one can utilize an iterative approach with a higher-order time difference representation.[39,43,44] Subiterations could also remove linearization and factorization errors. The iterative approach results in a 'multistage' implicit scheme. Research in multistage schemes coupled with the multigrid technique for unsteady computations may be interesting.[45]

Those implicit methods still have the stability limit in allowable time step sizes. Although the ADI method is not necessarily stable in three dimensions, the LU-SGS method is unconditionally stable. A main source of the limitation in the LU-SGS method could be the linearization. Improvements in the linearization may be necessary. Because of the complex formula of TVD upwind schemes, however, it is impractical to construct a second-order accurate noniterative implicit scheme using true Jacobians.[46]

Unsteady computations often require an order of magnitude more computational time than steady-state computations. For example, for the transonic flow computation about an oscillating wing,[41] a steady-state solution is necessary as an initial condition. A few cycles of unsteady computations are needed to obtain a periodic solution. At least two periodic solutions with different time step sizes are needed to verify the time accuracy. In total, more than an order of magnitude increase in time over a single steady-state computation is required. The development of an efficient time-accurate algorithm is highly desirable.

Turbulence Models

The Baldwin-Lomax model[47] is probably the most widely used model in the CFD community. It is simple and robust. It works fairly well for both attached and vortical flows. However, even for such a simple model, careful implementation is required: one constant is supposed to be $C_{wk} = 1.0\ (\neq 0.25)$ and the definition of $\vec{u}^2_{\max}$ $\left(\vec{u}^2_{DIF} = \vec{u}^2_{\max} - \vec{u}^2_{\min}\right)$ is supposed to be $\vec{u}_{\max} \equiv \vec{u}|_{F_{\max}}$ (not the maximum of $\vec{u}$; interested readers may refer to the original paper for the definitions of those variables).[48] Furthermore, for moving-grid cases, grid velocity should be subtracted from $\vec{u}$.

The 90's seemed to open with the eye-catching success of the Johnson-King model to simulate a transonic flow over the ONERA M6 wing.[49] Later, it was found that the result was "largely fortuitous" due to a coding error and a large dissipation by the CD scheme with the spectral-radius scalar dissipation.[50]

It is well known that turbulence modeling is the pacing item of CFD. Turbulence researchers tend to go up to higher-order closure models.[51] In aerodynamics, however, we don't usually need to know any of the turbulence statistical quantities. Coordinated efforts should be placed toward improving the lower-order models, such as Johnson-King and one-equation models.[52-54] In addition, the importance of unsteadiness should be addressed.

Code Validation for Unsteady Computations

To implement CFD in the design cycles for use in industry, it is very important to provide estimates of the accuracy of the numerical predictions. This led to a program of CFD validation at NASA.[5] Comparisons of the numerical results with experimental and other numerical data are the essence of the validation process. However, simple comparisons are not informative enough to properly validate CFD. A complete validation requires a careful study of all aspects of the numerical simulation.

CFD simulation consists of three modeling procedures: physical modeling, numerical modeling and geometric modeling. Physical modeling is the derivation of the governing equations including turbulence models. Numerical modeling is the algorithm used to solve the governing equations and boundary conditions, such as upwind schemes and numerical implementation of boundary conditions. Geometric modeling is the numerical representation of the geometry itself. Code validation needs to address each of the three modeling procedures.

Geometric modeling seems straightforward but could have significant effects on computed results. Let's consider a wing geometry. A computational grid is usually generated for a wing alone. Experimental model consists of the wing mounted on the tunnel wall and the other tunnel walls. Before making quantitative comparisons, we have to know the effects of differences in geometries. Furthermore, it is very common to modify the wing geometry because of the lack of data or simplicity for grid generation. For example, exact definition of the wing tip is often ignored for computational convenience. However, it could have a significant effect in the solutions.[55] More complicated model geometry may include minor differences at many locations. In addition, real models are not rigid. Model deformation in the experiment has to be considered.

Validating numerical modeling has a difficulty in defining grid quality. A universal measure is required to define the grid quality for various grids, topologies and zoning. Such measure should monitor local errors as well as global errors. Under a reasonable grid quality, grid convergence will validate the numerical modeling. As the grid and time-step sizes are refined, numerical solutions should converge asymptotically. The actual grid convergence may be difficult to show because of the cost and time limitations. At the very least the asymptotic behavior should be demonstrated. In addition, if the CD scheme is used, the amount of dissipation has to be checked carefully.

For physical modeling, the turbulence model is the most important pacing item because the governing equations of the fluid motion are well established. Performance of turbulence models may be coupled with numerical modeling as shown in Ref. 50. Geometric modeling is also important especially in the transonic regime. The boundary-layer transition is usually neglected as well. Thus, validation is difficult and time consuming and should proceed carefully.

After all those issues are addressed, we may still have a problem with validating against experiment. A major source of the remaining discrepancies could be facility limitations in the experiment. Importance of good experimentation should be emphasized for CFD validation.

Besides the uncertaintities of unsteady experiments,[4] comparison of unsteady computations with experiments has a difficulty as well. In steady state, comparison of surface pressures gives a certain measure of agreement. However, in an unsteady case, comparison of instantaneous pressures does not give a good measure because of the phase error that has to be considered. Therefore, computations of periodic flows are a logical step to extend CFD validation from steady to unsteady applications.

When the time history of local pressures shows sinusoidal variations, the pressures may be described by the first harmonic of a Fourier series:

$$p = p_s + \Delta p' \cos \omega t + \Delta p'' \sin \omega t = p_s + |\Delta p| \cos(\omega t - \varphi), \qquad (9)$$

where p and p_s denote the local and mean pressures, while $\Delta p'$ and $\Delta p''$ are the components of the harmonic.[56] Or $|\Delta p|$ and φ represents the magnitude and phase, respectively, of unsteady pressures. For transonic flows, particularly in the region of a shock wave, this expression is no longer true. In such cases higher harmonics may need to be added to the Fourier series. Despite those higher order harmonics, the first harmonic of a Fourier series gives a good measure of unsteady variations.

Toward the analysis of flutter, extensive research has been performed for transonic flows past oscillating wings in both CFD and experiment.[4,41,56-59] Coordinated validation efforts should be focused on such cases.

Recent Applications

This section reviews recent unsteady applications under NASA programs because they demonstrate the most advanced, computer intensive simulations. The major activities can be categorized into the Euler and Navier-Stokes computations. The latest Euler computations use the unstructured-grid approach, while the Navier-Stokes computations mostly use the structured-grid approach. The applications listed below also indicate the future trend toward the multidisciplinary approach.

Unstructured-Grid Euler Computations

A series of work has been done by a group at Langley Research Center toward aeroelastic simulations including flutter analysis.[60,61] An application of a hovering rotor has been done at Ames Research Center by using adaptive grid refinement.[62] Extension to dynamic adaption is ongoing.[63,64]

Structured-Grid Navier-Stokes Computations

• **Aeroelasticity:** A continuous effort has been performed by a group at Ames to develop aeroelastic computer codes.[41,65,66] The latest development couples shell finite-element structures with the Navier-Stokes solver.[67] Tail buffet at a high angle of attack has also been simulated.[68] Conventional flutter analysis coupled with the structured-grid, Euler/Navier-Stokes solver has also been done at Langley.[69]

• **Artificial Heart:** Computation of a flow through the artificial heart device is a significant spin-off of aerospace CFD done at Ames. The latest computation includes oscillating valves.[70]

• **Helicopter Rotors:** Extensive research of rotor aerodynamics has been performed at Ames, including acoustic propagation.[43,71,72] Recently, an overset grid approach has been investigated.[73] The unstructured-grid approach mentioned above is also a part of this group's activity. Assessment of turbulence models for highly separated flow is ongoing.[74]

• **SOFIA:** The Stratospheric Observatory For Infrared Astronomy (SOFIA) is a proposed successor of the existing airborne astronomical observatory, the Kuiper Airborne Observatory (KAO). A large cavity will be placed on the body of a Boeing 747SP for a telescope. A resonating cavity would endanger both the aircraft and telescope structure. Density fluctuations in the shear layer also cause optical fluctuation, which results in blurring of the telescope image. A numerical study has been performed at Ames to assess the cavity flow fields using the overset approach.[75]

• **Turbomachinery:** CFD application for turbomachinery is an area that had early success in unsteady computations at Ames.[7,76,77] Recent studies include grid adaptation using the structured-unstructured hybrid approach[78] and extension to acoustic analysis.[79]

• **Others:** Unsteady computations have been performed about a delta-wing in roll,[80] a wing-canard configuration undergoing pitching motion,[81] wing and wing-body configurations with oscillating control surfaces,[82,83] and a delta wing equipped with thrust reverser jets descending near the ground.[84] These studies are directing applications of CFD toward flight dynamics and controls.

Chemically Reacting Flow Computations

Although computations of chemically reacting flows are beyond the scope of this paper, unsteady computations have been done in this area. Interested readers may refer to Refs. 85-90.

Visualization

The ability to compute streak lines is essential for unsteady flow visualization because streak lines simulate experimental visualization. Such visualization has recently been performed at Ames by using the Unsteady Flow Analysis Tool (UFAT).[91] Other flow variables, such as density and pressure, are relatively easy to visualize because the unsteady flow animation can be generated from a sequence of instantaneous plots. However, visualizing unsteady results causes a major demand on disk storage. Even after some data reduction, several gigabytes of disk space is often required.

Concluding Remarks

This paper has reviewed several issues of CFD for unsteady aerodynamics. Basic algorithm issues and procedures necessary for code validation have been discussed. Sample applications indicate the trend of aerospace CFD research to multidisciplinary applications.

Toward its maturity as a scientific/engineering tool, software engineering will be a vital element of CFD.[92] Down selection of CFD codes is also inevitable to develop good, validated software. CFD researchers are accustomed to writing their own codes. These research codes are efficient in some sense because of the customization for specific needs. However, the codes do not communicate with each other and the data are not necessarily interchangeable. The codes often contain undocumented assumptions. The specific customization and assumptions of these codes make it very difficult for other users to tailor these codes for their needs and thus reduce overall efficiency. Furthermore, debugging and validating every research code will be overwhelming. Standardization is indispensable for CFD codes to be a solid tool. Efficiency is also important. Now, a new coordinated approach is required for CFD research.

On the other hand, the importance of CFD for aerospace engineering is growing. CFD will serve as a basic tool for multidisciplinary computational approaches that combine aerodynamics with structural dynamics, controls, and propulsion.[93] Such approaches require sustained teraFLOPS or faster computers that have massively parallel processors.

Acknowledgement

The author's work was supported by NASA Grant NCC 2-605. The author would like to thank G. S. Deiwert of NASA Ames Research Center for his suggestion about recent applications for chemically reacting flows.

References

[1] Chapman, D. R., "A Perspective on Aerospace CFD," *Aerospace America*, pp. 16-19, 58, January 1992.

[2] Jameson, A., "Numerical Wind Tunnel – Vision or Reality," AIAA Paper 93-3021, July 1993.

[3] MacCormack, R. W., "A Perspective on a Quarter Century of CFD Research," AIAA Paper-93-3291-CP, July, 1993.

[4] Nixon, D. (ed.), Unsteady Transonic Aerodynamics, Chapters 1, 3-6, Vol. 120, Progress in Astronautics and Aeronautics, AIAA, Washington DC., 1989.

[5] Marvin, J. G., "CFD Validation," Proceedings of the 5th International Symposium on Computational Fluid Dynamics – Sendai, Vol. II, pp. 178-185, August 1993.

[6] Kutler, P., "Multidisciplinary Computational Aerosciences," Proceedings of the 5th International Symposium on Computational Fluid Dynamics – Sendai, Vol. II, pp. 109-119, August 1993.

[7] Henne, P. A. (ed.), Applied Computational Aerodynamics, Chapters 3, 4, 13, 21, and 24, second printing, Vol. 125, Progress in Astronautics and Aeronautics, AIAA, Washington DC., 1990.

[8] Klopfer, G. H. and Molvik, G. A., "Conservative Multizone Interface Algorithm for the 3-D Navier-Stokes Equations," AIAA Paper 91-1601, June 1991.

[9] Applications of Mesh Generation to Complex 3-D Configurations, AGARD Conference Proceedings, No. 464, March 1990.

[10] Steger, J. L., "Technical Evaluation Report on the Fluid Dynamics Panel Specialists' Meeting on Applications of Mesh Generation to Complex 3-D Configurations," AGARD Advisory Report No. 268, March 1991.

[11] Special Course on Unstructured Grid Methods for Advection Dominated Flows, AGARD Report 787, May, 1992.

[12] Nakahashi, K. and Obayashi, S., "Viscous Flow Computations Using a Composite Grid," AIAA Paper 87-1128-CP, June 1987.

[13] Nakahashi, K., "External Viscous Flow Computations Using Prismatic Grid," Thirteenth International Conference on Numerical Methods in Fluid Dynamics, Lecture Notes in Physics 414, Springer-Verlag, pp. 280-284, 1992.

[14] Vinokur, M., "An Analysis of Finite-Difference and Finite-Volume Formulations of Conservation Laws," *Journal of Computational Physics*, Vol. 81, No. 1, pp. 1-52, March 1989.

[15] Thomas, P. D. and Lombard, C. K., "Geometric Conservation Law and Its Application to Flow Computations on Moving Grids," *AIAA Journal*, Vol. 17, No. 10, pp. 1030-1037, October 1979.

[16] Obayashi, S., "Freestream Capturing for Moving Coordinates in Three Dimensions," *AIAA Journal*, Vol. 30, No. 4, pp. 1125-1128, April 1992.

[17] Steger, J. L., "Implicit Finite Difference Simulation of Flow about Arbitrary Geometries with Application of Airfoils," *AIAA Journal*, Vol. 16, No. 7, pp. 679-686, July 1978.

[18] Jameson, A., Schmidt, W. and Turkel, E., "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, June 1981.
[19] Yee, H. C., Warming, R. F. and Harten, A., "Implicit Total Variation Diminishing (TVD) Schemes for Steady-State Calculations," AIAA Paper 83-1902-CP, July, 1983.
[20] Swanson, R. C. and Turkel, E., "Aspects of a High-Resolution Scheme for the Navier-Stokes Equations," AIAA Paper 93-3372-CP, July, 1993.
[21] Lomax, H., "CFD in the 1980's from One Point of View," AIAA Paper 91-1526-CP, June 1991.
[22] Van Leer, B., Thomas, J. L., Roe, P. L. and Newsome, R. W., "A Comparison of Numerical Flux Formulas for the Euler and Navier-Stokes Equations," AIAA Paper 87-1104-CP, June 1987.
[23] Roe, P. L., "Approximate Riemann Solvers, Parameter Vectors, and Difference Schemes," *Journal of Computational Physics*, Vol. 43 pp. 357-372, 1981.
[24] Fujii, K. and Obayashi S., "High-Resolution Upwind Scheme for Vortical-Flow Simulations," *Journal of Aircraft*, Vol. 26, No. 12, pp. 1123-1129, December 1989.
[25] Peery, K. M. and Imlay, S. T., "Blunt-Body Flow Simulations," AIAA Paper 88-2904, 1988.
[26] Roberts, T. W., "The Behaviour of Flux Difference Splitting Schemes near Slowly Moving Shock Waves," *Journal of Computational Physics*, Vol. 90, pp. 141-160, 1990.
[27] Quirk, J. J., "A Contribution to the Great Riemann Solver Debate," ICASE Report No. 92-64, November 1992.
[28] Einfeldt, B., Munz, C. D., Roe, P. L. and Sjögreen, B., "On Godunov-Type Methods near Low Densities," *Journal of Computational Physics*, Vol. 92 pp. 273-295, 1991.
[29] Wada, Y., "An Improvement of the HLLEM Scheme and Its Extension to Chemically Reacting Flows," Second U.S. National Congress on Computational Mechanics, Washington, D.C., August 1993, as well as NAL TR-1189, 1993.
[30] Liou, M-S, "On a New Class of Flux Splittings," Thirteenth International Conference on Numerical Methods in Fluid Dynamics, Lecture Notes in Physics 414, Springer-Verlag, pp. 115-119, 1992.
[31] Öksüzoğlu, H., "State Vector Splitting for the Euler Equations of Gasdynamics," AIAA Paper 92-0326, January, 1992.
[32] Gooch, C. F. and Öksüzoğlu, H., "Extension of State-Vector Splitting to the Navier-Stokes Equations," AIAA Paper 93-3374, July, 1993.
[33] Van Leer, B., "Progress in Multi-Dimensional Upwind Differencing," Thirteenth International Conference on Numerical Methods in Fluid Dynamics, Lecture Notes in Physics 414, Springer-Verlag, pp. 1-26, 1992.
[34] Catalano, L. A., De Palma, P., Napolitano, M. and Pascazio, G., "A Critical Analysis of Multi-Dimensional Upwinding for the Euler Equations,"

Proceedings of the 5th International Symposium on Computational Fluid Dynamics – Sendai, Vol. I, pp. 35-40, August 1993.
[35] Steinhoff, J., Senge, H. and Yonghu, W., "Computational Vortex Capturing – Reduction of Numerical Diffusion," Proceedings of the 4th International Symposium on Computational Fluid Dynamics – Davis, Vol. III, pp. 93-100, September 1991.
[36] Felici, H. M. and Drela, M., "An Eulerian/Lagrangian Coupling Procedure for Three-Dimensional Vortical Flows," AIAA Paper 93-3370-CP, July, 1993.
[37] Rai, M. M. and Chakravarthy, S., "Conservative High-Order-Accurate Finite-Difference Methods for Curvilinear Grids," AIAA Paper 93-3380-CP, July, 1993.
[38] Pulliam, T. H. and Steger, J. L., "Recent Improvement in Efficiency, Accuracy, and Convergence for Implicit Approximate Factorization Algorithms," AIAA Paper 85-360, January 1985.
[39] Pulliam, T. H., "Time Accuracy and the Use of Implicit Methods," AIAA Paper 93-3360-CP, July, 1993.
[40] Obayashi, S., Matsushima, K., Fujii, K. and Kuwahara, K., "Improvements in Efficiency and Reliability for Navier-Stokes Computations using the LU-ADI Factorization Algorithm," AIAA Paper 86-0338, January 1986.
[41] Obayashi, S., Guruswamy, G. P., and Goorjian, P. M., "Streamwise Upwind Algorithm for Computing Unsteady Transonic Flows past Oscillating Wings," *AIAA Journal*, Vol. 29, No. 10, pp. 1668-1677, October 1991. Errata: *AIAA Journal*, Vol. 30, No. 2, pp. 569, February 1992.
[42] Yoon, S. and Kwak, D., "Implicit Navier-Stokes Solver for Three-Dimensional Compressible Flows," *AIAA Journal*, Vol. 30, No. 11, pp. 2653-2659, November 1992.
[43] Chen, C. L., McCroskey, W. J. and Obayashi, S., "Numerical Solutions of Forward-Flight Rotor Flow Using an Upwind Method," *Journal of Aircraft*, Vol. 28, No. 6, pp. 374-380, June 1991.
[44] Matsuno, K., "Higher-Order Time-Accurate Scheme for Unsteady, Three-Dimensional Flows," AIAA Paper 93-3362-CP, July, 1993.
[45] Arnone, A., Liou, M-S. and Povinelli, L. A., "Multigrid Time-Accurate Integration of Navier-Stokes Equations," AIAA Paper 93-3361-CP, July, 1993.
[46] Venkatakrishnan, V., "Viscous Computations Using a Direct Solver," Computers & Fluids, Vol. 18, No. 2, pp. 191-204, 1990.
[47] Baldwin, B. S. and Lomax, H., "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, January 1978.
[48] Kaynak, Ü., "Turbulence Modeling for Applied Computational Aerodynamics," Proceedings of the 4th International Symposium on Computational Fluid Dynamics – Davis, Vol. I, pp. 576-581, September 1991.
[49] Abid, R., Vatsa, V. N., Johnson, D. A. and Wedan, B. W., "Prediction of Separated Transonic Wing Flows with Nonequilibrium Algebraic Turbulence Model," *AIAA Journal*, Vol. 28, No. 8, pp.1426-1431, August, 1990.

[50] Rumsey, C. L. and Vatsa, V. N., "A Comparison of the Predictive Capabilities of Several Turbulence Models Using Upwind and Central-Difference Computer Codes," AIAA Paper 93-0192, January 1993.
[51] Hytopoulos, E. and Simpson, R. L., "Critical Evaluation of Recent Second-Order Closure Models," AIAA 93-0081, January 1993.
[52] Johnson, D. A., "Nonequilibrium Algebraic Turbulence Modeling Considerations for Transonic Airfoils and Wings," AIAA Paper 92-0026, January 1992.
[53] Baldwin, B. S. and Barth, T. J., "A One-Equation Turbulence Transport Model for High Reynolds Number Wall-Bounded Flows," AIAA Paper 91-0610, January, 1991.
[54] Spalart, P. R. and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," AIAA Paper 92-0439, January 1992.
[55] Srinivasan, G. R., McCroskey, W. J., Baeder, J. D. and Edwards, T. A., "Numerical Simulation of Tip Vortices of Wings in Subsonic and Transonic Flows," AIAA Journal Vol. 26, No. 10, pp. 1153-1162, October 1988.
[56] Tijdeman, H. and Seebass, R., "Transonic Flow past Oscillating Airfoils," *Annual Review of Fluid Mechanics*, x8157, Vol. 12, pp. 181-222, 1980.
[57] Transonic Unsteady Aerodynamics and Aeroelasticity, AGARD Conference Proceedings 507, March 1992.
[58] Simpson, L. B. and Whitfield, D. L., "Flux-Difference Split Algorithm for Unsteady Thin-Layer Navier-Stokes Solutions," *AIAA Journal*, Vol. 30, No. 4, pp. 914-922, April 1992.
[59] Hixon, R., Tsung, F-L. and Sankar L. N., "A Comparison of Two Methods for Solving 3-D Unsteady Compressible Viscous Flows," AIAA Paper 93-0537, January 1993.
[60] Batina, J. T., "Implicit Upwind Solution Algorithms for Three-Dimensional Unstructured Meshes," *AIAA Journal*, Vol. 31, No. 5, pp. 801-805, 1993.
[61] Rausch, R. D., Batina, J. T. and Yang, H. T. Y., "Three-Dimensional Time-Marching Aeroelastic Analyses Using an Unstructured-Grid Euler Method," *AIAA Journal*, Vol. 31, No. 9, pp. 1626-1633, September 1993.
[62] Strawn, R. C. and Barth, T. J., "A Finite-Volume Euler Solver for Computing Rotary-Wing Aerodynamics on Unstructured Meshes," *Journal of American Helicopter Society*, Vol. 38, No. 2, pp. 61-67, April 1993.
[63] Rausch, R. D., Batina, J. T. and Yang, H. T. Y., "Spatial Adaptation Procedures on Tetrahedral Meshes for Unsteady Aerodynamic Flow Calculations," AIAA Paper 93-0670, January 1993.
[64] Biswas, R. and Strawn, R., "A New Procedure for Dynamic Adaption of Three-Dimensional Unstructured Grids," AIAA Paper 93-0672, January 1993.
[65] Guruswamy, G. P., "Vortical Flow Computations on a Flexible Blended Wing-Body Configuration," *AIAA Journal*, Vol. 30, No. 10, pp. 2497-2503, October 1992.
[66] Obayashi, S. and Guruswamy, G. P., "Unsteady Shock-Vortex Interaction on a Flexible Delta Wing," *Journal of Aircraft*, Vol. 29, No. 5, pp. 790-798, September-October 1992.

[67] Guruswamy, G. P. and Byun, C., "Fluid-Structure Interactions Using Navier-Stokes Flow Equations Coupled with Shell Finite Element Structures," AIAA Paper 93-3087, July, 1993.
[68] Rizk, Y. M., Guruswamy, G. P. and Gee, K., "Numerical Investigation of Tail Buffet on F-18 Aircraft," AIAA Paper 92-2673, June 1992.
[69] Lee-Rausch, E. M. and Batina, J. T., "Calculation of AGARD Wing 445.6 Flutter Using Navier-Stokes Aerodynamics," AIAA Paper 93-3476, August 1993.
[70] Kiris, C. Rogers, S., Kwak, D. and Chang, I-D., "Computation of Incompressible Viscous Flows through Artificial Heart Devices with Moving Boundaries," Fluid Dynamics in Biology, Contemporary Mathematics, Vol. 141, American Mathematical Society, Providence, 1993.
[71] Srinivasan, G. R., Baeder, J. D., Obayashi S. and McCroskey, W. J., "Flowfield of a Lifting Rotor in Hover: A Navier-Stokes Simulation," *AIAA Journal*, Vol. 30, No. 10, pp. 2371-2378, October 1992.
[72] Srinivasan, G. R. and Baeder, J. D., "TURNS: A Free-Wake Euler/Navier-Stokes Numerical Method for Helicopter Rotors," *AIAA Journal*, Vol. 31, No. 5, pp. 959-962, May 1993.
[73] Meakin, R. L., "Moving Body Overset Grid Methods for Complete Aircraft Tiltrotor Simulation," AIAA Paper 93-3350, July 1993.
[74] Srinivasan, G. R., Ekaterinaris, J. A. and McCroskey, W. J., "Dynamic Stall of an Oscillating Wing Part 1: Evaluation of Turbulence Models," AIAA Paper 93-3403, August 1993.
[75] Atwood, C. A., "Selected Computations of Transonic Cavity Flows," 1993 ASME Fluids Engineering Conference, Forum on Computational Aero- an Hydro-Acoustics, Washington, D. C., June 1993.
[76] Rangwalla, A. A., Madavan, N. K. and Johnson, P. D., "Application of an Unsteady Navier-Stokes Solver to Transonic Turbine Design," *Journal of Propulsion and Power*, Vol. 8 No. 5, pp. 1079-1086, September-October 1992.
[77] Gundy-Burlet, K. L., Rai, M. M. and Madavan, N. K., "Unsteady, Three-Dimensional, Navier-Stokes Simulations of Multistage Turbomachinery Flows," AIAA Paper 93-1979, June 1993.
[78] Mathur, S. R., Madavan, N. K. and Rajagopalan, R. G., "A Solution-Adaptive Hybrid-Grid Method for the Unsteady Analysis of Turbomachinery," AIAA 93-3015, July, 1993.
[79] Rangwalla, A. A. and Rai, M. M., "Computational Analysis of Tonal Acoustics Arising from Rotor-Stator Interaction in Turbomachines," Proceedings of International Noise and Vibration Control Conference, "Noise - 93", Vol. 5, pp. 261-268, May 1993.
[80] Chaderjian, N. M. and Schiff, L. B., "Navier-Stokes Prediction of a Delta Wing in Roll with Vortex Breakdown," AIAA Paper 93-3495, August 1993.
[81] Tu, E. L., Obayashi, S. and Guruswamy, G. P., "Unsteady Navier-Stokes Simulation of the Canard-Wing-Body Ramp Motion," AIAA Paper 93-3058, July 1993.
[82] Klopfer, G. H. and Obayashi, S., "Virtual Zone Navier-Stokes Computations for Oscillating Control Surfaces," AIAA Paper 93-3363-CP, July 1993.

[83] Obayashi, S., Chiu, I-T. and Guruswamy, G. P., "Navier-Stokes Computations on Full-Span Wing-Body Configuration with Oscillating Control Surfaces," AIAA Paper 93-3687, August 1993.
[84] Chawla, K. and Van Dalsem, W. R., "Numerical Simulation of a Powered-Lift Landing," Paper No. 32, AGARD 72nd Fluid dynamics Panel Meeting and Symposium on Computational and Experimental Assessment of Jets in Cross Flow, Winchester, U. K., April 1993.
[85] Cambier, J. L., Adelman, H. G. and Menees, G. P., "Numerical Simulation of a Pulsed Detonation Wave Augmentor Device," AIAA paper 93-1985, June, 1993.
[86] Polsky S. and Cambier, J. L., "Numerical Study of the Transient Flow in the Driven Tube and the Nozzle Section of a Shock Tunnel," AIAA Paper 93-2018, June, 1993.
[87] Sussman, M. A., "Source Term Evaluation for Combustion Modeling," AIAA Paper 93-0239, January 1993.
[88] Wilson, G. J. and Sussman M. A., "Computation of Unsteady Shock Induced Combustion Using Logarithmic Species Conservation Equations," *AIAA Journal*, Vol 31, No. 2, pp. 294-301, February 1993.
[89] Wilson, G. J., "Time-Dependent Quasi-One Dimensional Simulations of High Enthalpy Pulse Facilities," AIAA Paper-92-5096, December 1992.
[90] Wilson, G. J., "Time-Dependent Simulations of Reflected-Shock/Boundary Layer Interaction," AIAA Paper-93-0480, January 1993.
[91] Lane, D. A., "Visualization of Time-Dependent Flow Fields," IEEE Visualization '93, San Jose, October 1993.
[92] Rizzi, A., Eliasson, P., Lindblad, I., Hirsch, C., Lacor, C. and Haeuser, J., "The Engineering of Multiblock/Multigrid Software for Navier-Stokes Flows on Structured Meshes," Computers & Fluids, Vol. 22 No. 2/3, pp. 341-367, 1993.
[93] Holst T. L., Salas, M. D. and Claus, R. W., "The NASA Computational Aerosciences Program – Toward TeraFLOPS Computing," AIAA Paper 92-0558, January 1992.

REPORT DOCUMENTATION PAGE	Form Approved OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
November 1993

3. REPORT TYPE AND DATES COVERED
Contractor Report

4. TITLE AND SUBTITLE
Progress in Computational Unsteady Aerodynamics

5. FUNDING NUMBERS
NCC2-605

6. AUTHOR(S)
Shigeru Obayashi

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
MCAT Institute
3933 Blue Gum Drive
San Jose, CA 95127

8. PERFORMING ORGANIZATION REPORT NUMBER
A-94021

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA CR-177630

11. SUPPLEMENTARY NOTES
Point of Contact: Guru P. Guruswamy, Ames Research Center, MS 258-1, Moffett Field, CA 94035-1000; (415) 604-6329

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified — Unlimited
Subject Category 02

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

After vigorous development for over twenty years, Computational Fluid Dynamics (CFD) in the field of aerospace engineering has arrived at a turning point toward maturity. This paper discusses issues related to algorithm development for the Euler/Navier-Stokes equations, code validation and recent applications of CFD for unsteady aerodynamics. Algorithm development is a fundamental element for a good CFD program. Code validation tries to bridge the reliability gap between CFD and experiment. Many of the recent applications also take a multidisciplinary approach, which is a future trend for CFD applications. As computers become more affordable, CFD is expected to be a better scientific and engineering tool.

14. SUBJECT TERMS
Unsteady aerodynamics, Aeroelasticity, Computational Fluid Dynamics (CFD)

15. NUMBER OF PAGES
23

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified		

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102