

N91-210634

THOUGHTS ON THE CHIMERA METHOD OF SIMULATION
OF THREE-DIMENSIONAL VISCOUS FLOW2707
P-10

Joseph L. Steger

Department of Mechanical, Aeronautical, and Materials Engineering
University of California at Davis
Davis, California 95616

cc 907079

ABSTRACT

The chimera overset grid method is reviewed and discussed relative to other procedures for simulating flow about complex configurations. It is argued that while more refinement of the technique is needed, current schemes are competitive to unstructured grid schemes and should ultimately prove more useful.

INTRODUCTION AND OVERVIEW

There are currently two mainstream approaches for computing flow fields in which geometry imposes complex boundary conditions – composite structured grid schemes and unstructured grid schemes. In my assessment of the literature, unstructured grid methods are generally considered to be more versatile and easier to adapt to complex geometry while composite structured grid methods are generally considered to use more efficient numerical algorithms and require less computer memory. But with either composite-structured or unstructured grids, the capability to solve flow about complex configurations has been suitably demonstrated. Both pure-strain approaches have their strengths and weaknesses. Hybrid schemes which incorporate the best features of both have already appeared[1-3] and will likely become more important in treating flow about complex geometries.

The chimera[4] and similar methods which use overset grids[5-29] are generally classed into the composite structured grid category, because these approaches clearly grew out of an attempt to generalize body conforming structured grid schemes to treat more complex situations. The chimera approach uses a composite of overset structured grids to resolve geometry, flow features, or permit more efficient flow solvers. While chimera generally employs composite structured grids, the connectivity of the overset structured grids is itself unstructured.

The chimera approach has been used to compute inviscid and high Reynolds flow about complex configurations(c.f.[7,8,14-19,24-26]), and it has even been demonstrated for unsteady three dimensional viscous flow problems in which one body moves with respect to another[26]. The viability of this approach is perhaps best illustrated by the fact that this progress has been made by a relatively small group of researchers. Nevertheless, chimera is sometimes viewed as an intermediate

solution approach, one which will ultimately be replaced by the unstructured grid method. With further examination, however, the chimera approach may be found to have more versatility than current unstructured schemes because, while grids can be abutted together like patches, they can also be overset. Oversetting can be somewhat foreign to finite volume and finite element methods, but oversetting can be useful. In overset schemes, intermediate boundary curves can be placed arbitrarily. In overset regions, the possibility also exists of impressing solutions from one domain onto another via forcing functions rather than only through boundary interfaces. These features can and have been exploited in several ways.

Overset grids allow structured grids to be used without excessive distortion or inefficient use of grid density. Consequently, efficient numerical methods can be used which depend on structured grids such as alternating direction implicit schemes and parabolized Navier Stokes procedures. Moreover, numerical schemes that use structured-grids generally require less computer storage and are better suited to vectorized computers. Arbitrary placement of intermediate boundaries can greatly simplify the task of structured grid generation. It allows, for example, the use of hyperbolic grid generation procedures[30,31] which do not conform to boundary value constraints but which generate nearly orthogonal grids with excellent mesh spacing control. Overset structured grids have also been used as a solution adaptation procedure[21,22], and overset grids can be positioned in the field simply to implement a special solver, most of which require some kind of coordinate alignment, (usually to streamlines in flow field simulation). Overset grids allow one body to move with respect to another without regriding[10,26] or placement of new bodies into the domain without regenerating the entire mesh.

As noted earlier, the chimera method is an outgrowth of trying to generalize a powerful solution approach, the body conforming structured grid method, to more complex situations. The method is proven, but far from mature. There are weaknesses which must be removed if chimera is to remain competitive with unstructured grids.

There are two main criticisms leveled against the current implementations of the chimera method. One is the bookkeeping-like complexities associated with connecting overset grids together. In fact, the bookkeeping with a chimera scheme is similar to that associated with an unstructured grid – easier because the connectivities can be made using structured grid data but more difficult because of multiple oversetting. However, only a few researchers have worked on this problem as opposed to the myriad that have worked on unstructured grids. Consequently, there is less available software for overset structured grids.

The other criticism is that in most simulations of complex flows the solutions on the overset grids are merged using simple interpolation. The fact that interpolation is generally used to connect grids implies that conservation is not strictly enforced. For most practical applications it is difficult to devise a situation where this error is overall-significant since conservation is strictly maintained at all points in the domain (assuming the solver is a conservative one) except at a small number of interface boundaries. Nevertheless, this source of error has to be eliminated, and conservative interface schemes for overset grids have been devised[23,27]. Refinement and simplification of these techniques are still warranted for use with three dimensional flow solvers, however.

In the remainder of this paper, a brief review of the chimera scheme is given, and a few results from previously presented space shuttle flow field simulations are used to indicate current status. Some possible future directions for the chimera scheme are then indicated, followed by concluding remarks.

BACKGROUND AND PROGRESS

The chimera composite grid discretization method is a domain decomposition approach which uses overset body-conforming grids. In this approach, each component of a configuration is grided separately and is overset onto a major grid to form the complete model. The major grid is stretched over the entire field, and is often generated about a dominant boundary or body surface. Minor grids are used to resolve features of the geometry or flow that are not adequately resolved by the major grid, and are overset on the major grid without requiring mesh boundaries to join regularly.

For example, Fig. 1 shows surface grids generated for the integrated space shuttle configuration in its ascent mode. The configuration shown has simplified attach hardware, and various protuberances such as the external fuel lines and even the orbiter vertical tail have been neglected. A grid is then independently generated about each component. A composite grid is then formed by superimposing all grids together. The body-conforming grids used for each component are shown in Fig. 2 at their respective planes of symmetry. Here the external tank (ET) grid is treated as the major grid and is extended to the far field. Figure 3 shows nearby $\xi = \text{constant}$ planes for the orbiter and ET projected onto an $x = \text{constant}$ plane. Whenever points of a grid, say grid 1, fall within the body-boundary of another grid, say grid 2, the points of grid 1 are cut out forming a hole in grid 1. The hole-boundary data of grid 1 are then supplied from grid 2. Hole grid points have been removed from view in Fig. 3.

Software to interconnect the grids is needed to ascertain when points of one grid fall within a body boundary of another (grid hole points) and to supply pointers so that one grid can provide boundary data to another. Various algorithms have been devised for performing these tasks automatically [4,8,14-16,20,29]. For the illustrated shuttle grids, the code Pegasus [8,14,16,29] (provided and maintained by CALSPAN of AEDC) has been used. General software for this problem has also been developed in [20] and includes an interactive workstation demonstrator for two dimensional grids.

A flow simulation code developed for a single general curvilinear grid can be readily adapted for composite overset grids. One simply sets flags to blank out hole points and supplies a control program that calls in grids and interface routines. For example, the structured-grid, implicit, approximately-factored F3D scheme [32,33] for the thin-layer Navier-Stokes equations

$$\partial_\tau \hat{Q} + \partial_\xi \hat{F} + \partial_\eta \hat{G} + \partial_\zeta \hat{H} = Re^{-1} \partial_\zeta \hat{S} \quad (1)$$

was easily modified for chimera overset grids as

$$\begin{aligned} & \left[I + i_b h \delta_\xi^b (\hat{A}^+)^n + i_b h \delta_\zeta \hat{C}^n - i_b h Re^{-1} \bar{\delta}_\zeta J^{-1} \hat{M}^n J - i_b h D_i |_\zeta \right] \\ & \times \left[I + i_b h \delta_\xi^f (\hat{A}^-)^n + i_b h \delta_\eta \hat{B}^n - i_b h D_i |_\eta \right] \Delta \hat{Q}^n = \\ & - i_b \Delta t \{ \delta_\xi^b (\hat{F}^+)^n + \delta_\xi^f (\hat{F}^-)^n + \delta_\eta \hat{G}^n + \delta_\zeta \hat{H}^n - Re^{-1} \bar{\delta}_\zeta \hat{S}^n + (D_e |_\eta + D_e |_\zeta) \hat{Q}^n \} \end{aligned} \quad (2)$$

Here introduction of the flag i_b accommodates the possibility of having arbitrary holes in the grid. (The hole includes hole-boundary points which are later updated by interpolating the solution from the overset grid which created the hole.) The array of values i_b is defined such that $i_b = 1$ at normal grid points and $i_b = 0$ at hole points. Thus, in Eq. (2) when $i_b = 1$ the normal scheme

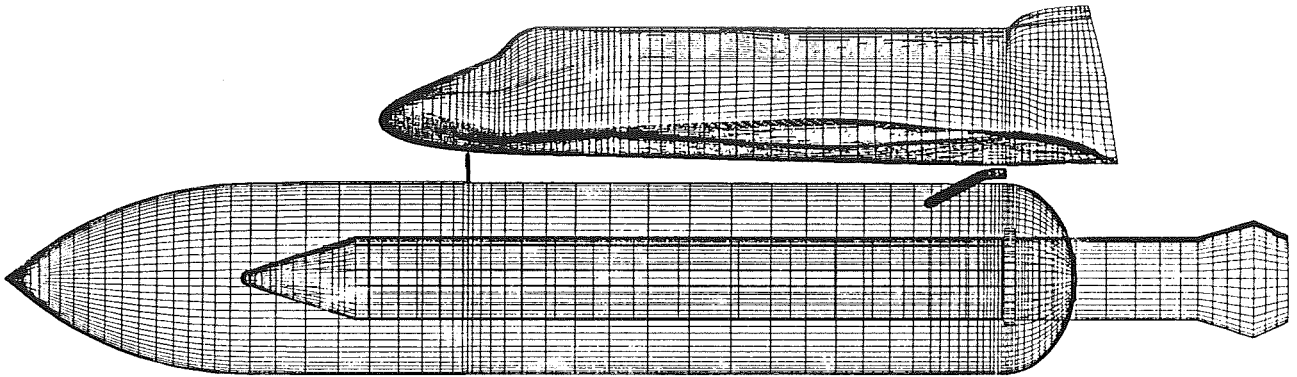


Fig. 1 Simplified configuration and surface grid point distributions for the integrated space shuttle.

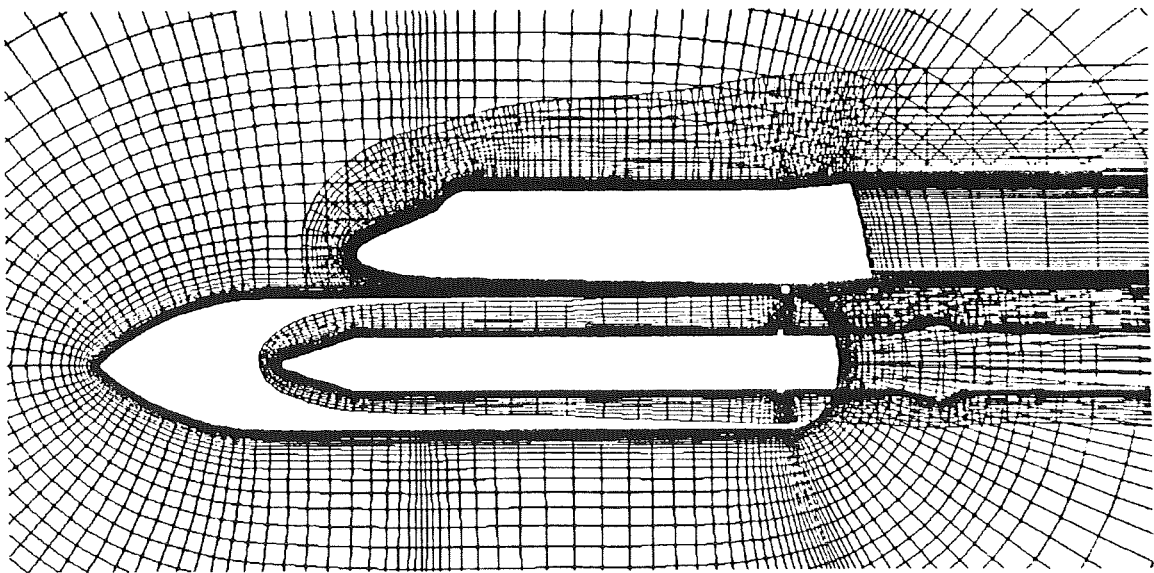


Fig. 2 Symmetry planes of all grids.

is maintained, but when $i_b = 0$ the scheme reduces to $\Delta\hat{Q}^n = 0$ or $\hat{Q}^{n+1} = \hat{Q}^n$ and thus \hat{Q} is not changed at a hole point. For the most part i_b is coupled to the time step ($h = \Delta t$ or $(\Delta t)/2$) and is trivial to implement into the coding. (Difference operators that use more than one point to either side require some additional coding modification, see Ref. [24]) By using the i_b array it is not necessary to provide branching logic to avoid hole points, and computer vectorization is not inhibited.

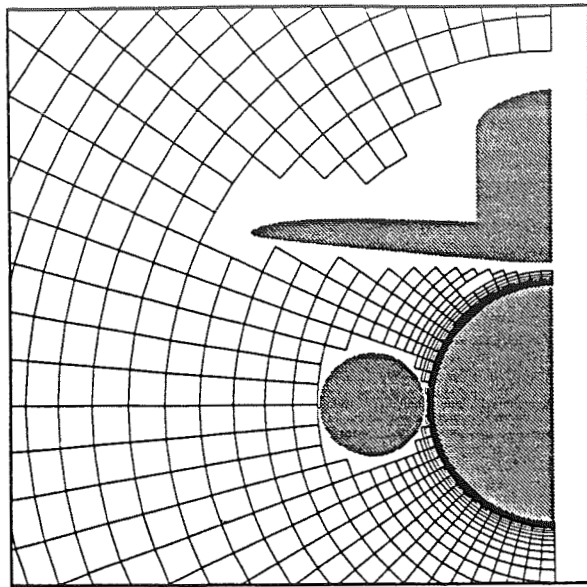
The F3D algorithm was implemented on composite grids by writing a control program which at each "time step" fetched a grid and its data from an isolated large memory into a working memory. Boundary interface arrays that store grid interconnect data, Q_{BC} , are also fetched. The Q_{BC} array holds overset-grid boundary values for the current grid which are supplied from the other grids and is a relatively small array. Because the hole boundaries are arbitrarily located, the Q_{BC} array has pointers much like those used with unstructured grids. The solution on the current grid is then updated or advanced in time. Overset boundary data that the current grid sends to other grids are then found by interpolation and loaded into Q_{BC} , and all arrays are sent back to the isolated large memory. The next grid is then fetched, and so on.

To illustrate this capability, calculated results for the integrated space shuttle vehicle are taken from Refs. [24,25] and are reproduced in Figs. 4 and 5. These figures show comparisons between computational and experimental data for $M_\infty = 1.05$ at an angle of attack, $\alpha = -3^\circ$, and using the wind tunnel Reynolds number $Re = 4.0 \times 10^6/\text{ft}$ for the computations. Shaded surface pressure coefficient comparisons between the computation and wind tunnel data [34] are shown by Fig. 4. This kind of comparison is possible because the 3% scale wind tunnel model was instrumented with 1538 pressure taps. Mach contours in planes of symmetry of the ET and solid rocket booster (SRB) are also shown in Fig. 4, and are used to highlight the SRB plume which was modeled as a hot-air jet. A limited amount of flight test data [35] are also available for comparison, and Fig. 5 shows pressure comparisons between computation, flight, and wind tunnel data taken along the side of the fuselage. This computation required about 15 hours of computer time using a single processor of the CRAY 2, and employed a composite grid containing one million points distributed over seven distinct grids. Additional details of the experimental comparisons (and some disclaimers) are given in Refs.[24,25].

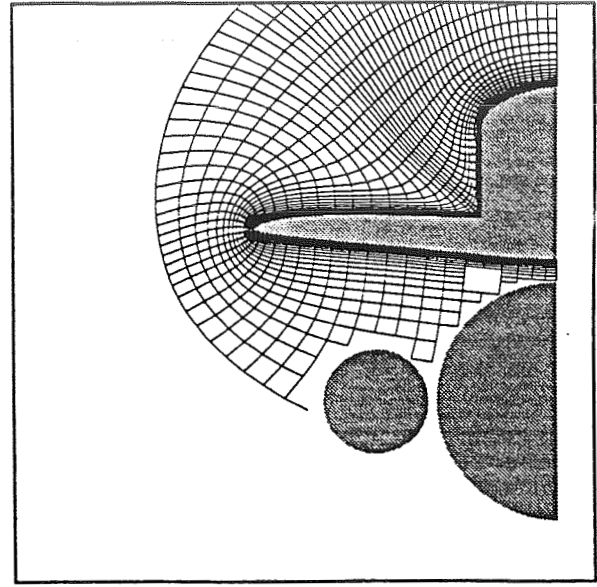
With the composite overset grid approach it is possible to move one body with respect to another without regriding at each time step advance of the flow field. Demonstration of this capability for SRB staging from the shuttle is presented in Ref. [26] .

FUTURE DIRECTIONS

As noted previously, two main criticisms can be leveled against the chimera approach: 1) the complexity of the interconnectivity is perhaps as difficult as dealing with an unstructured grid, and 2) nonconservative interpolations to update interface boundaries are often used in practical three dimensional computations. To indicate the complexity of the interconnectivity, it should be remarked that some of the attach-hardware used in the space shuttle simulations is not actually attached. The attach-hardware was floated between the body elements. This is because the algorithms devised to impose grid connectivity are not accurate enough when the refined grids used for



a) ET grid



b) orbiter grid

Fig. 3 Cross-section of grids showing holes, a)ET grid, b)Orbiter grid.

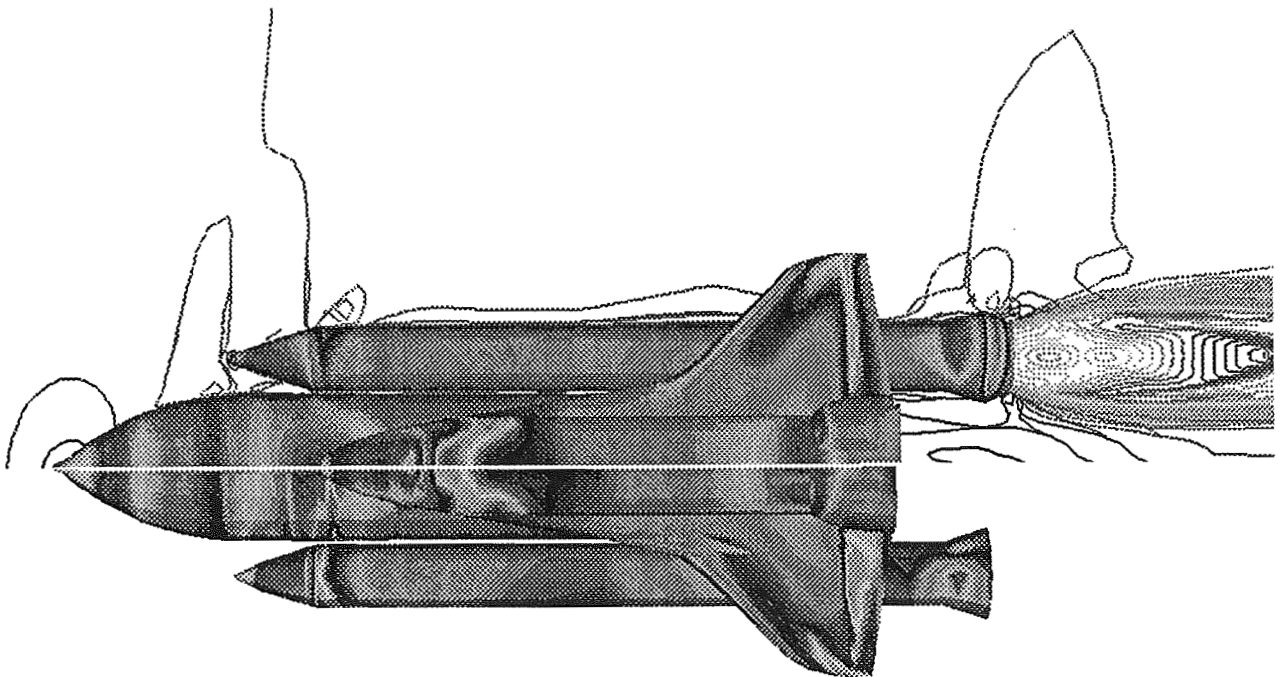


Fig. 4 Comparison of pressure coefficient between computation (top) and wind tunnel (bottom), $M_\infty = 1.05$, $\alpha = -3^\circ$, and $Re = 4.0 \times 10^6/ft$ (3% model).

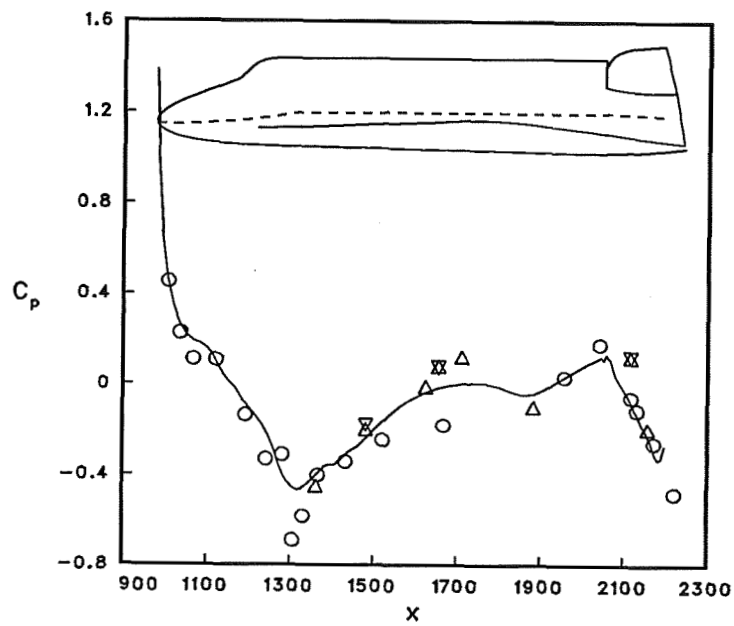


Fig. 5 Comparison of C_p from computation (-), wind tunnel (o), and flight test (∇ right side, Δ left side) along the $\phi = 70^\circ$ line of the orbiter fuselage, $M_\infty = 1.05$ and $\alpha = -3^\circ$, $Re = 4.0 \times 10^6/ft$, and $10^\circ/9^\circ$ elevon deflection.

high Reynolds number viscous flow simulation are encountered. Because the normal grid spacing is so fine, small errors in interpolation for the body surface can cause the test which identifies when points lie inside a another body surface to fail. Specifically, two viscous boundary layer grids emanating off the same surface cannot fall within that surface, yet due to the fineness of the grids, a viscous flow field point above the surface of one grid may be judged to be inside the same body surface of another grid because of interpolation error. Special logic can be used to exclude this case, and the problem is not encountered for inviscid simulations. There are a variety of ways to deal with this problem, they include more consistent interpolation schemes, introducing special viscous surface grids, alternate tests, and so on, but current software is inadequate and needs refinement.

Several approaches have been proposed to treat the problem of nonconservative interfaces, but they have not been implemented into the F3D code. It should be remarked that while the space shuttle simulations have used simple interpolation procedures because of their robustness, CALSPAN simulations for the last several years[16-18,28,29] have implemented an unpublished idea of Benek and use interpolants of delta quantities, specifically, $\hat{Q}^{n+1} - \hat{Q}^n$. Interpolating this quantity on interface boundaries ensures space-time conservation over the global field, but the utility (or penalty) of this approach has not yet been rigourously examined.

A fairly obvious way to ensure interface conservation would be to introduce an unstructured flow solver in the vicinity of the interface boundaries. Already in chimera, primitive elements of an unstructured grid solver exist in the form of pointers and grid interfacing arrays which transfer interpolated values of the solution from one grid to the next. Some care would have to be taken, but an explicit differencing of the governing equations using unstructured data could substitute for the interpolation process. The chimera would then mimic a hybrid structure-unstructured approach much as in Refs. [1-3].

Finally, the chimera framework lends itself to the construction of a general-purpose flow code that can optionally take advantage of approximate solution methods, and some preliminary work has been carried out in this area. For example, the F3D flow simulation code used for the previously described shuttle work already has options (at various levels of maturity) to use either explicit or implicit solution algorithms as well as a semimarching scheme for predominately steady supersonic flow. In addition, a fortified option [36] of the basic algorithm is available. To support this option and to provide diagnostics of Navier-Stokes solution accuracy, a three-dimensional boundary-layer code in arbitrary general coordinates has also been included.

CONCLUDING REMARKS

Overset grid schemes such as chimera grew out of efforts to adapt body conforming structured grid methods to more complex boundaries. Although these schemes have not received the attention that unstructured (or patched) grid methods have received, they have proven to be competitive, and are likely to prove to be more powerful. Incorporation of unstructured grids into the overall chimera framework is quite feasible, and offers a relatively low risk route to a hybrid structured-unstructured simulation code as well as a fall back position in the (unlikely) event that chimera does not live up to its expectations.

REFERENCES

1. Nakahashi, K., "FDM-FEM Zonal Approach for Computations of Compressible Viscous Flows". 10th International Conference on Numerical Methods in Fluid Dynamics. Beijing, China, June, (1986).
2. Nakahashi, K. and Obayashi, S., "Viscous Flow Computations Using a Composite Grid", AIAA Paper No. 87-1128-CP, AIAA 8th Computational Fluid Dynamics Conference, (1987).
3. Weatherill, N. P., "On the Combination of Structured-Unstructured Meshes", in *Numerical Grid Generation in Computational Fluid Dynamics*, S. Sengupta, ed., Pineridge Press, (1988).
4. Steger, J. L., Dougherty, F. C., and Benek, J. A., "A Chimera Grid Scheme", in *Advances in Grid Generation*, K.N. Ghia and U. Chia, eds., ASME FED-5, 59-69, (1983).
5. Starius, G., "Composite Mesh Difference Methods for Elliptic and Boundary Value Problems", *Numeri. Math.* Vol. 28, (1977).
6. Atta, E. H., "Component Adaptive Grid Interfacing". AIAA Paper 81-0382, (1981).
7. Atta, E. H. and Vadyak, J. A., "A Grid Interfacing Zonal Algorithm for Three-Dimensional Transonic Flows about Aircraft Configurations". AIAA Paper 82-1017, (1982).
8. Benek, J. A., Buning, P. G., and Steger, J. L., "A 3-D Grid Embedding Technique", AIAA Paper No. 85-1523, (1985).
9. Eberhardt, D. S., "Overset Grids in Compressible Flow", AIAA Paper 81-0382, (1985).
10. Dougherty, F. C., Benek, J. A., and Steger, J. L., "On Application of Chimera Grid Scheme to Store Separation", NASA TM 88193, (1985).
11. Henshaw, W. D., "Part I: The Numerical Solution of Hyperbolic Systems of Conservation Laws; Part II: Composite Overlapping Grid Techniques", Ph.D. thesis, California Institute of Technology, (1985).
12. Chesshire, G., "Composite Grid Construction and Applications", Ph.D. thesis, California Institute of Technology, (1986).
13. Lombard, C.K.; and Venkatapathy, E., "Implicit Boundary Treatment for Joined and Disjoint Patched Mesh Systems". AIAA Paper 85-1503-CP, (1985).
14. Benek, J. A., Steger, J. L., Dougherty, F. C. and Buning P. G., "Chimera: A Grid Embedding Technique", AEDC-TR-85-64, Arnold Air Force Station, TN, (1986).
15. Steger, J. L. and Benek, J. A., "On the Use of Composite Grid Schemes in Computational Aerodynamics", *Comp. Methods Appl. Mech. and Eng.*, vol. 64, nos. 1-3, (1987).
16. Benek, J. A., Donegan, T. L., and Suhs, N. E., "Extended Chimera Grid Embedding Scheme with Application to Viscous Flows". AIAA-87-1126-CP, (1987).
17. Suhs, N. E., "Computations of Three-Dimensional Cavity Flow at Subsonic and Supersonic Mach Numbers", AIAA 87-1208, (1987).
18. Donegan, T. L., Benek, J. A., and Erickson, J. C., "Calculation of Transonic Wall Interference", AIAA 87-1432, (1987).
19. Van Dalsem, W. R., Panaras, A. G., and Steger, J. L., "Numerical Investigation of a Jet in Ground Effect with a Crossflow", SAE 872344, from P-203 - Proceedings of the International Powered Lift Conference, Santa Clara, CA. (1987).
20. Brown, D. L., Chesshire, G., and Henshaw, W. D., "Getting Started With CMPGRD", Center for Computational Mathematics and Mechanics, The Royal Institute of Technology, S-100 44 Stockholm, Sweden, (1988).

21. Berger, M.J., "Adaptive Mesh Refinements for Hyperbolic Partial Differential Equations", STAN-CS-82-924, Stanford University, (1982).
22. Berger, M.J. and Olinger, J., "Adaptive Mesh Refinements for Hyperbolic Partial Differential Equations", J. Comp. Physics, Vol. 53, (1987).
23. Berger, M.J., "On Conservation at Grid Interfaces", ICASE Report 84-43, NASA Langley Research Center, Hampton, Virginia, (1984).
24. NASA Ames Space Shuttle Flow Simulation Group: P. G. Buning, I. T. Chiu, S. Obayashi, Y. M. Rizk, and J. L. Steger, "Numerical Simulation of the Integrated Space Shuttle Vehicle in Ascent", AIAA Atmospheric Flight Mechanics Conference, Minneapolis, (1988).
25. NASA Ames Space Shuttle Flow Simulation Group: P. G. Buning, I. T. Chiu, F. W. Martin Jr., R. L. Meakin, S. Obayashi, Y. M. Rizk, J. L. Steger, and M. Yarrow, "Flowfield Simulation of the Space Shuttle Vehicle in Ascent", Proceedings of the Fourth International Conference on Supercomputing, Santa Clara, CA, (1989).
26. Meakin, R. L. and Suhs, N., "Unsteady Aerodynamic Simulation of Multiple Bodies in Relative Motion", AIAA 9th Computational Fluid Dynamics Conference to be held at Buffalo, NY, (1989).
27. Moon, Y. J. and Liou, M.-S., "Conservative Treatment of Boundary Interfaces for Overlain Grids and Multi-level Grid Adaptations", AIAA Paper 89-1980-CP, (1989).
28. Fox, J. and Allee, E., "Experimental/Computational Study of a Transonic Aircraft with Stores". AIAA Paper 89-1832, (1989).
29. Dietz, W. E., Jacocks, J. L., and Fox, J. H., "Application of Domain Decomposition to the Analysis of Complex Aerodynamic Configurations", SIAM Conference on Domain Decomposition Methods, Houston, TX, (1989).
30. Steger, J. L. and Rizk, Y. M., "Generation of Three Dimensional Body-Fitted Coordinates Using Hyperbolic Partial Differential Equations". NASA TM 86753, (1985).
31. Eisenman, P. R. and Erlebacher, G., "Grid Generation for the Solution of Partial Differential Equations", ICASE Report No. 87-57, (1987).
32. Ying, S. X., Steger, J. L., Schiff, L. B., and Baganoff, D., "Numerical Simulation of Unsteady, Viscous, High Angle of Attack Flows Using a Partially Flux-Split Algorithm". AIAA Paper No. 86-2179, (1986).
33. Steger, J. L., Ying, S. X., and Schiff, L.B., "A Partially Flux-Split Algorithm for Numerical Simulation of Compressible Inviscid and Viscous Flow". *Proceedings of the Workshop on Computational Fluid Dynamics*, Institute of Nonlinear Sciences, U. Calif., Davis, (1986).
34. Spangler, R. H., "Results of Tests Using a 0.03 Scale Model (47-OTS) of the Space Shuttle Integrated Vehicle in the AEDC 16 Foot Transonic Propulsion Wind Tunnel (IA105A)", NASA CR 160851, Vol.2, (1981).
35. Rockwell International Internal Letter SAS/AERO/83-094, "STS-5 Postflight Report on Ascent External Aerodynamic Loads", March (1983).
36. Van Dalsem, W. R. and Steger, J. L., "Using the Boundary-Layer Equations in Three-Dimensional Viscous Flow Simulation", AGARD Conference Proceedings No. 412, Aix-en-Provence, France, (1986).