SUMMARY OF RESEARCH REPORT

NASA Research Grant NAG-1-1587

UNSTRUCTURED MESH METHODS FOR THE SIMULATION OF HYPERSONIC FLOWS

February 1, 1994 – March 31, 1999

Technical Monitor: K.L. Bibb
NASA Langley Research Center
Mail Stop 408A
Hampton, VA 23665-5225

Principal Investigator: J. Peraire
Professor, Aeronautics and Astronautics
Massachusetts Institute of Technology
Room 37-451
Cambridge, MA 02139

Submitted July, 1999

SUMMARY

This report summarizes the research undertaken, at Aeronautics Department of the Massachusetts Institute of Technology, during the approximately five year period, February 94 – March 99, under NASA Grant NAG-1-1587. This work is part of a larger effort aimed at providing a reliable fast turn around capability for the prediction of hypersonic flows over complete vehicle configurations.

RESEARCH CONTRIBUTIONS

Previous to this grant the principal investigator had been involved, also under NASA sponsorship, in the development of the unstrucutred mesh system FELISA for the calculation of inviscid transonic flows over complete aircraft configurations. One of the major objectives of the current grant was to evolve the technology within FELISA with the main objective being the development of a robust practical tool that could be used in a production environment and could successfully handle hypersonic viscous re-entry flows.

This grant was funded in five different phases covering the following periods:

Phase I :	2/1/94	-	31/5/95
Phase II :	1/6/95	-	31/5/96
Phase III :	1/6/96	-	31/7/97
Phase IV :	1/8/97	-	31/7/98
Phase V :	1/8/98	-	3/31/99

During this grant period effort concentrated on the described below. For each heading we indicate in brackets the period during which most of this task took place.

1. Mesh Generation

1.1 Delaunay Volume Generator [I,II]

The volume generator within the FELISA system was based on the advancing front algorithm. For grids of small size (under a million elements) this generator was adequate, but for larger grids, such as those required for the more recent calculations, the generator was slow and the time spent in volume discretization was comparable to that spent in computing the flow solution. For this reason a much faster, more reliable algorithm, based on a combination of Delaunay and advancing front method ideas was developed. The developed generator was about ten times faster that the existing generator within FELISA and is still being used extensively.

1.2 Viscous Mesh Generation [III,IV]

Some research was devoted to extend the existing unstructured grid generation algorithms to the automatic generation of viscous meshes. A method for generating directionally refined unstructured tetrahedral methods was proposed. The motivation was the need to efficiently mesh the complex computational domains which are frequently encountered in aerospace applications. The proposed method is applicable to the construction of meshes suitable for the simulation of inviscid and viscous aerodynamic flows. Minimum user intervention is required and the user specified stretching distribution is achieved by locally modifying an existing mesh. The stretched tetrahedra generated are such the largest angle occurring on any tetrahedron is close to ninety degrees. The directional refinement is driven by a scalar function which measures the distance to a prescribed curve or surface. Features such as leading and trailing edges can be efficiently discretized in this manner, thus making directional refinement a useful feature in the generation of inviscid meshes.

1.3 CEM/CSMGrid Generation [III]

Realistic problems in computational electromagnetics and computational structural mechanics involve geometrical features, such as wires, shells, multi materials, etc., which require special attention from the mesh generation point of view. Since the generator developed originally for aerodynamic applications found a wider user base, both within and outside NASA, some effort was devoted to incorporate the ability to handle non-manifold geometries. The developed generator had the ability to handle multiple volumes, with common surfaces, wires and zero thickness walls. In addition, procedures for the easy identification of the elements, edges and triangles forming each geometric feature have also been incorporated.

1.4 Parallel Mesh Generation [III,IV]

In order to reduce mesh generation times and to avoid the very large single-processor requirements in sequential generators, some has been carried out into developing parallel mesh generation algorithms. A strategy capable of generating a grid in parallel was proposed. The algorithm is based on a sequence of point insertion and load balancing cycles. Points are inserted concurrently within each subdomain and dynamic load-balancing is incorporated to ensure an even load distribution during the generation process. The mesh inter-processor boundaries are allowed to move to accommodate element migration between subdomains. The strategy proposed is capable of generating a grid in parallel such that when the grid generation process is finished, the grid is already partitioned and in place for the flow solution.

1.5 Native Geometry, MGEN and CAPRI libraries [IV,V]

The objective here is to develop a general-purpose tool (GRIDEX) for geometry manipulation and unstructured mesh generation for aerodynamic applications. The tool should allow for the for the direct use of boundary representation models generated using different CAD packages, automatic topology specification, and interactive manipulation of the geometry necessary for mesh generation applications. In order to carry out all of the above tasks in the same tool a modularized system was proposed and an initial prototype was developed. The main modules of the proposed system are:

A Native Geometry Module. Although geometry creation is largely done using third party CAD software, some modifications are usually required in order for this geometry to be useful for CFD analyses, e.g. repairing or adding additional surfaces to define the far field boundaries. The purpose of developing this module is therefore to provide some basic means for defining additional geometry such as far field boundaries, or wakes, required for the grid generation process. The module develop to date is very simple and uses cubic and bi-cubic curves and surfaces; similar to the current capability within the FELISA grid generator.

Computational Analysis Programming Interface (CAPRI). The main purpose of this module is to provide a common geometry interface between the different third party geometry packages and the application programs such as grid generators and solvers. Under this grant the specific interface of CAPRI with IDEAS was developed. Interfaces with other systems such as Parasolids or Pro-Engineer were already available.

Library of Unstructured Mesh Generation Routines (MGEN). The main grid generation algorithms within the FELISA system have been further enhanced and re-coded in the form of a library can be easily interfaced with the other libraries. The main calls to this library carry out tasks such as edge, surface and volume discretization.

The two additional modules required to construct the initial prototype, a Graphics Library and the Driver Program were develop under separate projects at the NASA Langley GeoLab.

2. Flow Solution

2.1 Hypersonic Flow Solver [I,II,IV]

Work here concentrated on the development of a hypersonic finite volume flow solver which is adequate for hypersonic flows. It was found that the techniques developed, primarily for subsonic and transonic flows, failed to give accurate and stable solutions for hypersonic flows, especially in three dimensions. In order to produce stable and accurate hypersonic computations, two developments were necessary: 1) The use of dissipation forms which strictly enforce monotonicity at shocks and 2) Identifying Riemann solvers that do not suffer from after shock instability problems at high Mach numbers.

The dissipation model adopted is valid for general unstrucutred meshes and is adequate for hypersonic flows and at the same time produces accurate results in the transonic flow regime. The viscosity model is based on the Local Extremum Diminishing (LED) ideas proposed by Jameson.

The after shock instabilities, usually referred to as the "carbuncle phenomenon", were found to be more pronounced in unstructured meshes than in structured meshes. These instabilities appear as a result of undamped shear modes in some Riemann solvers. The study carried out considered cost and accuracy of various Riemann solvers, and identified the flux splitting methods of Haenel, Liou-Steffen and Harten-Lax-van Leer-Eindfelt as viable alternatives for use with unstructured meshes.

Real gas effects are essential in some hypersonic flow applications. A version of the flow solver, which incorporates real gas effects, was developed at NASA and support to carry out these modifications was provided through this grant.

Some research was also conducted into enhancing the capabilities of the current algorithms to reduce the losses caused by the numerical truncation error. An approach based on the replacement of the streamwise momentum equation, in smooth flow regions, by the enforcement of conservation of total pressure was proposed. Some initial test carried out in 2D were found to be encouraging.

2.2 Parallel Flow Solver [III]

The flow solver was parallelized, using a message passing strategy, and a fair amount of effort was devoted into optimizing its parallel efficiency. In particular points and edges within each processor were grouped into those that require communication and those that don't. This subdivision allows the computations to be organized in such a way that there is a significant overlap between computation and communication. The mesh partitioning algorithm is based on coordinate bisection and the MPI message passing protocol was employed to ensure portability. Very good scalability of the code was demonstrated. The parallel flow solver developed has been used extensively within NASA.

2.3 Finite Element Flow Solver [V]

Some effort was devoted into trying to assess the merits of Finite Element based algorithms for the problems of interest in this project. While only 2D test were performed during the period of this grant the results were very encouraging and further effort will be spent on developing these algorithms as a possible alternative to traditional finite volume methods. We were able to successfully write and test a finite element SUPG algorithm, which utilized both linear and quadratic elements and observed optimal convergence in both cases. A new pre-conditioner for use with symmetrizing entropy variables was proposed, which was able to resolve supersonic, transonic and subsonic flows up to the incompressible limit. The use of higher order elements combined with an implicit Newton solution technique is identified as one the most promising approaches on which to base a viscous flow solver. This line of research is one of the main thrusts in the current follow-on research program.

4. Aerodynamic Design [II,III,IV]

A methodology for performing optimization of three dimensional unstructured grids based on the Euler equations was presented. An efficient adjoint formulation was implemented as it allows situations involving a large number of design variables to be considered. This is a typical requirement in most realistic aerodynamic applications. The same, low-memory explicit relaxation algorithm is used to resolve the discrete equations that govern the flow, linearized direct, and adjoint problems. Mesh movement is performed in such a way that optimization of arbitrary geometries is allowed. Design using surface target pressure was demonstrated as well as multi-point lift-constrained drag minimization for complex aircraft geometries.

5. Error estimation and Adaptivity

The requirements of a numerical simulation are twofold: the approximation must be sufficiently coarse so as to permit repeated appeal within the design context; the approximation must be sufficiently fine so that the numerical prediction of the desired outputs, such as drag or lift, is representative of the true performance of the system. A posteriori error analysis offers great promise in reconciling these often conflicting requirements. A posteriori error analysis is composed of two critical ingredients: an estimation procedure which inexpensively assesses the error in a particular numerical solution ; and an adaptive refinement procedure which optimally exploits this error information to optimally improve the numerical solution. The objectives of a posteriori error control are the elimination of the numerical uncertainty, and to improve numerical efficiency.

We have proposed an a posteriori finite element error estimation methodology which has been applied to a number of both linear and non-linear problems including the steady incompressible Navier-Stokes equations. Our approach provides rigorous sharp, inexpensive bounds for the engineering outputs of interest. The potential advantages of such a methodology in the computation of compressible flows could be dramatic. One of the main stumbling blocks in extending this methodology to compressible flows is being able to handle discontinuous solutions. This topic is the focus of much of our current research.

REFERENCES

Below we give a list of publications generated as a result of the research developed under this grant.

MESH GENERATION

J. PERAIRE, and K. MORGAN, 'Unstructured mesh generation including directional refinement for aerodynamic flow simulation', Finite Elements in Analysis & Design, 25, p. 343-356, 1997.

T.OKUSANYA and J.PERAIRE, 'Parallel unstructured mesh generation', Proc. 5th International Conference on Numerical Grid Generation, Mississippi, April 1996.

T. OKUSANYA and J. PERAIRE, '3D Parallel unstructured mesh generation', to be presented at the 1997 Joint ASME/ASCE/SES Summer Meeting, June 29-July 2, 1997, Norris Center, Northwestern University.

J. PERAIRE, and K.MORGAN, 'Automatic generation of unstructured meshes for Navier Stokes flows', AIAA Paper 98-3010, Albuquerque, NM, June, 1998.

FLOW SOLVER

K.L. BIBB, J. PERAIRE, and C.J. RILEY, 'Hypersonic flow computations on unstructured meshes', Paper AIAA 97-0625, 1997.

K.MORGAN and J.PERAIRE, 'Unstructured grid finite element methods for fluid dynanics', Reports on Progress in Physics, Volume 61, Number 6, June 1998, Pages 569-638. Published by the Institute of Physics.

M.DRELA, A.MERCHANT, and J. PERAIRE, 'Elimination of spurious loss in Euler equation computations', AIAA Paper 98-2424, Albuquerque, NM, June, 1998.

K.E.WILLCOX and J.PERAIRE, 'Aeroelastic computations in the time domain using unstructured meshes', Int. J. Num. Meth. in Engng.,Vol. 40, 2413-1431, 1997.

AERODYNAMIC DESIGN

J. ELLIOT and J. PERAIRE, 'Aerodynamic design using unstructured meshes', Paper AIAA 96-1941, 1996.

J. ELLIOT and J. PERAIRE, 'Practical 3D aerodynamic design and optimization using unstructured meshes', Proc. of the 6th AIAA/NASA/USAF Multidisciplinary Analysis & Optimization Symp., Seattle, September 1996. Also, Paper AIAA 96-4170.

J. ELLIOT, and J. PERAIRE, 'Practical 3D aerodynamic design and optimization using unstructured meshes', AIAA J., Vol. 35, No. 9, September 1997.

J. ELLIOT, and J. PERAIRE, 'Aerodynamic optimizationon unstructured meshes with viscous effects', AIAA Paper 97-1849, Snowmass, CO. June 1997.

J.ELLIOTT and J.PERAIRE, 'Constrained, multipoint shape optimization for complex 3D configurations', submitted to the Aeronautical Journal, Nov. 1997.

J. ELLIOT, and J. PERAIRE, 'Progress towards a 3D aerodynamic shape optimization tool for the compressible high-Re Navier-Stokes equations discretized on unstructured meshes', AIAA Paper 98-2897, Albuquerque, NM, June, 1998.

ERROR ESTIMATION

M. PARASCHIVOIU, J. PERAIRE, and A.T. PATERA, 'A posteriori finite element bounds for linear functional output of elliptic partial differential equations', Comput. Methods Appl. Mech. Engrg., 150 p. 289-312, 1997.

J. PERAIRE and A.T. PATERA, 'Bounds for linear-functional outputs of coercive partial differential equations: Local indicators and adaptive refinement', Proc. of the Workshop On New Advances in Adaptive Computational Methods in Mechanics, Elsevier, Ed. P.Ladeveze and J.T. Oden, Cachan, Sept. 17-19, 1997.

J.PERAIRE and A.T.PATERA, 'Asymptotic a Posteriori finite element bounds for the outputs of noncoercive problems: the Helmholtz and Burgers equations', to appear in Comp. Meth. Appl. Mech. Engrg., Dec. 1997.

L.MACHIELS, A.T.PATERA, J.PERAIRE, and Y.MADAY, 'A general framework for finite element a posteriori error control: Application to linear and nonlinear convection-dominated problems', presented at ICFD Conference on Numerical Methods for Fluid Dynamics, Oxford, England, March 31-April 3, 1998.

L.MACHIELS, J.PERAIRE, and A.T.PATERA, 'Output bound approximations for incompressible Navier-Stokes equations', Proc. of the Istanbul Workshop on Industrial and Environmental Applications of Direct and Large Eddy Numerical Simulation, August, 1998.