N91-10893

3-D UNSTRUCTURED GRIDS FOR THE SOLUTION OF THE EULER EQUATIONS

C. Gumbert

NASA Langley Research Center

P. Parikh, S. Pirzadeh

Vigyan Research Associates

R. Löhner

George Washington University

Advancing Front Grid Generator

The advancing front technique is being used to develop a code to generate grids around complex three dimensional configurations for use in computing the invisid flow solutions by the Euler equations. By the advancing front technique points are introduced concurrently with the connectivity information so that a separate library is not required. The generation of a 3–D grid is accomplished in several steps. First the boundaries of the domain to be gridded must be described by two-, three– or four-sided surface patches. Next, a background mesh is required to control the grid spacing and stretching throughout the domain. This coarse tetrahedral grid is not required to conform to any of the boundaries. Next, each of the patches is mapped to 2–D, triangulated by the advancing front technique and mapped back to 3–D. These triangles form the initial front for the generation of the final tetrahedral mesh.

ADVANCING FRONT GRID GENERATOR

- The field points are introduced *while* grid is generated.

  - Does not require a separate library to introduce points before grid generation.

  - Allows adaptive regeneration of grids easily.

- Generation Steps:

  - Define boundaries of the domain using surface patches.

  - Set up background grid to control local grid characteristics.

 - 2-D triangulation of patches

  ⇒ Initial Front

  - Advancing front in 3-D to fill region with tetrahedra.

Advancing Front in 2-D

The figure is a schematic showing the advancing front technique in 2-D. The initial front is a set of line segments called faces. In 3-D a face is a triangle. In succession, each face of the front is deleted and a new point in the field is introduced. As in the center figure, if the new point is close to an existing point, then the existing one is used instead. The front advances until it closes in on itself at which point the region is fully gridded.

ADVANCING FRONT IN 2-D

deleted face

initial front
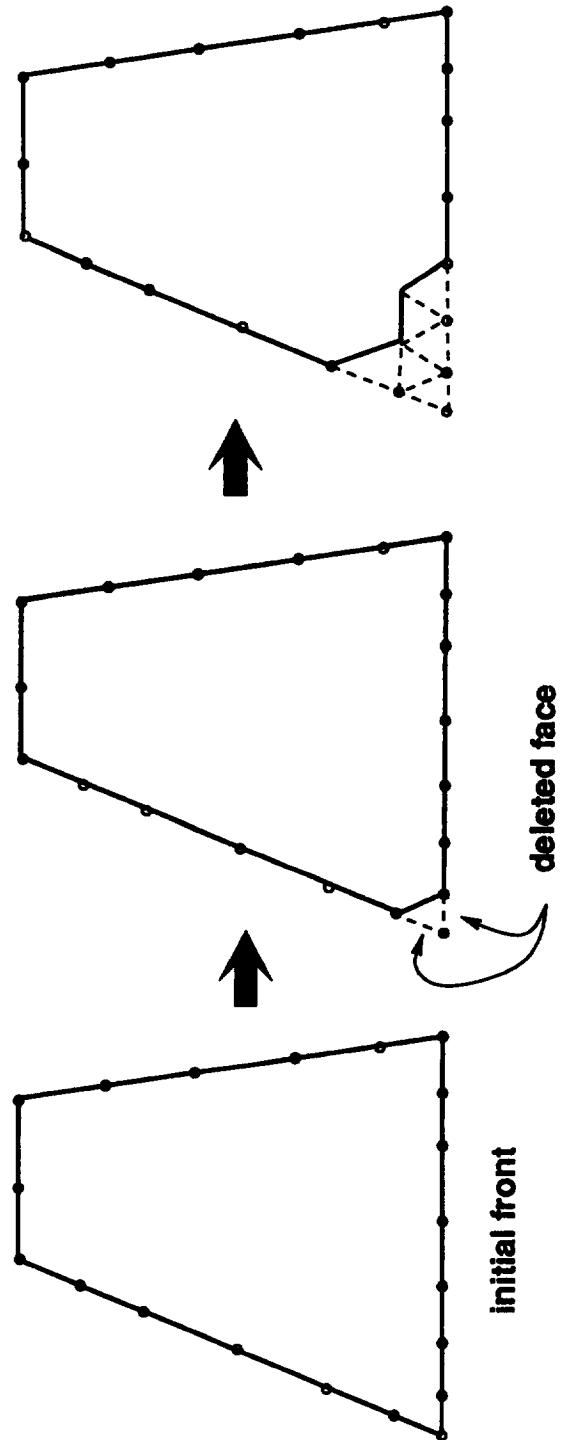
Surface Grid for F-18 Configuration

The figure shows the surface grid for an F-18 fighter configuration. The grid was generated for only one half of the configuration but it has been mirrored in the picture. The grid consists of 367,000 tetrahedrons using nearly 66,000 points of which 10,000 lie on the 75 surface patches defining the airplane and the computational box. It required about 400 seconds on the NAS CRAY-2 computer.
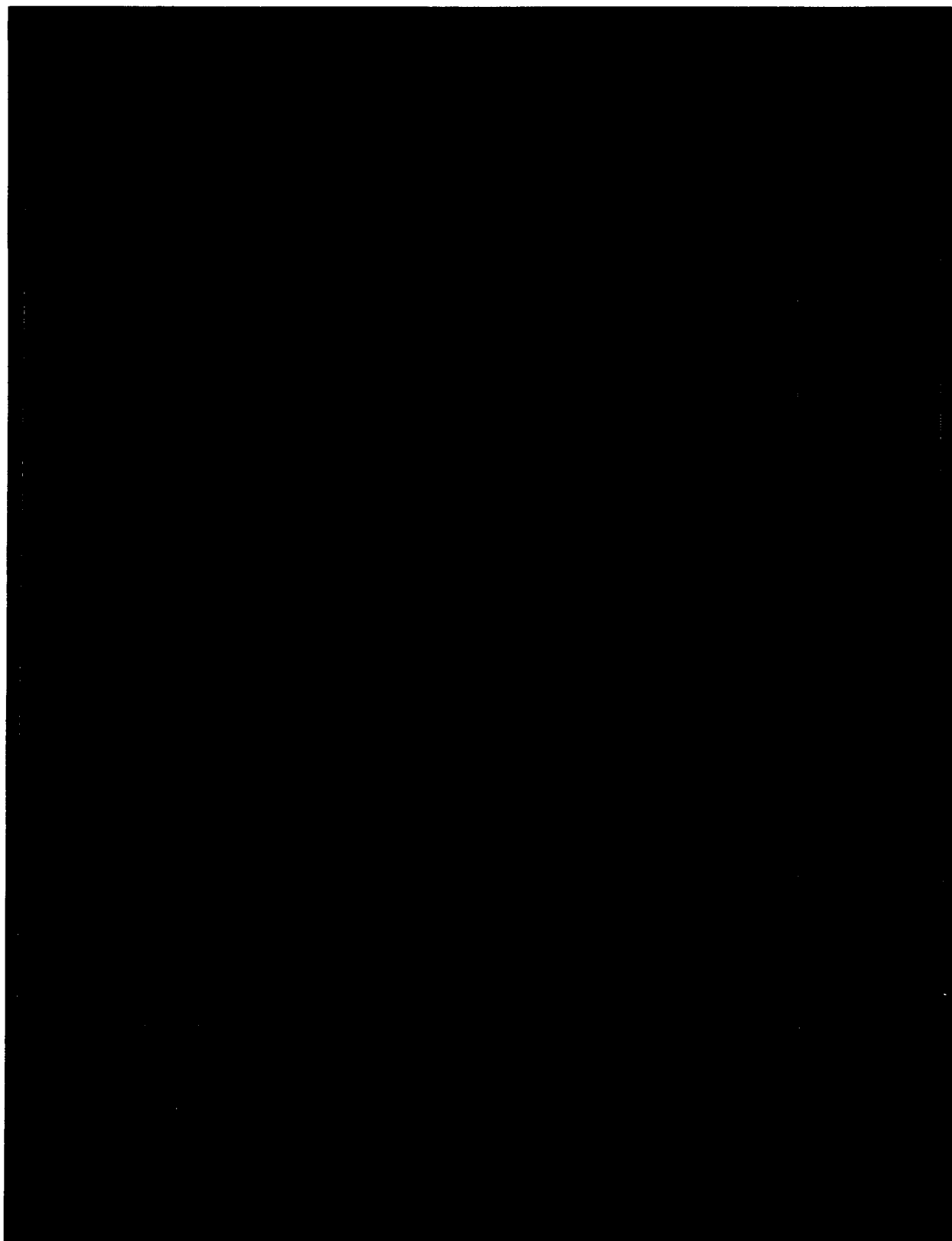
Finite Element Flow Solver

The flow solver developed in conjunction with the grid generator uses a two-step Taylor Galerkin finite element method for the Euler equations cast in Arbitrary Lagrangian-Eulerian (ALE) form. The Galerkin weighted residual method is used to perform the spatial discretization. Timestepping options are available for steady-state or transient problems. Accurate solutions without spurious over/undershoots can be obtained using Flux Corrected Transport (FCT) techniques. Second order pressure or Lapidus damping is used near shocks. Adaptive mesh refinement is used to better capture sharp gradients. This technique has not been fully implemented in 3-D.

FINITE ELEMENT FLOW SOLVER

- Two-step Taylor-Galerkin scheme

- Spatial discretization performed
  via Galerkin weighted residual
  method using linear elements

- Options for:

  Global/local timestepping

  Second order pressure or Lapidus damping

  Flux Corrected Transport (FCT)

  Adaptive H-refinement

Results In 2–D

As a means of determining the accuracy of this code, results are presented for two 2–D flow calculations. The first case is the transonic flow calculation for an NACA 0012 airfoil at $M_\infty = .80$ and $\alpha = 1.25°$. Comparison is made with results from FLO52, a finite volume method calculated on an O-type grid.

The second case is the flow field around a 20° ramp at $M_\infty = 3.0$. The exact solution is known and can be compared. This case also illustrates the use of adaptive mesh refinement.

RESULTS IN 2–D

- **Comparison of NACA 0012 airfoil**

    at $M_\infty = .80, \quad \alpha = 1.25°$

    FEFLO27 (unstructured mesh, Finite Element solver)

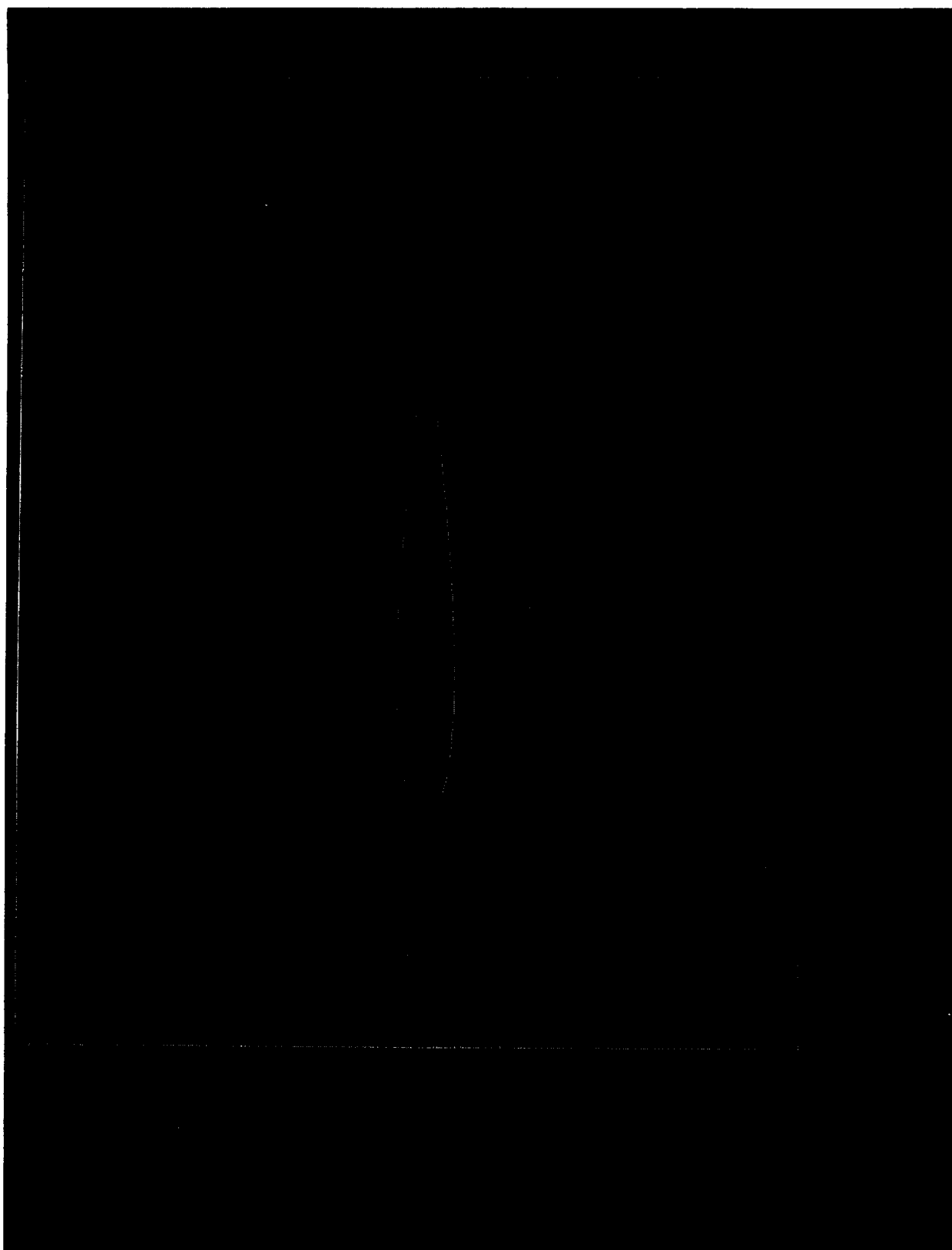   FLO52 (O-grid, finite volume method)

- **$20°$ ramp flow**

    at $M_\infty = 3.0$

   adaptive mesh refinement

Grid for NACA 0012 Airfoil

The figure shows the unstructured grid used for the calculation. There are nearly 5000 points in the grid of which 266 points lie on the airfoil surface. This is similar to the 193 points in the structured grid of FLO52.

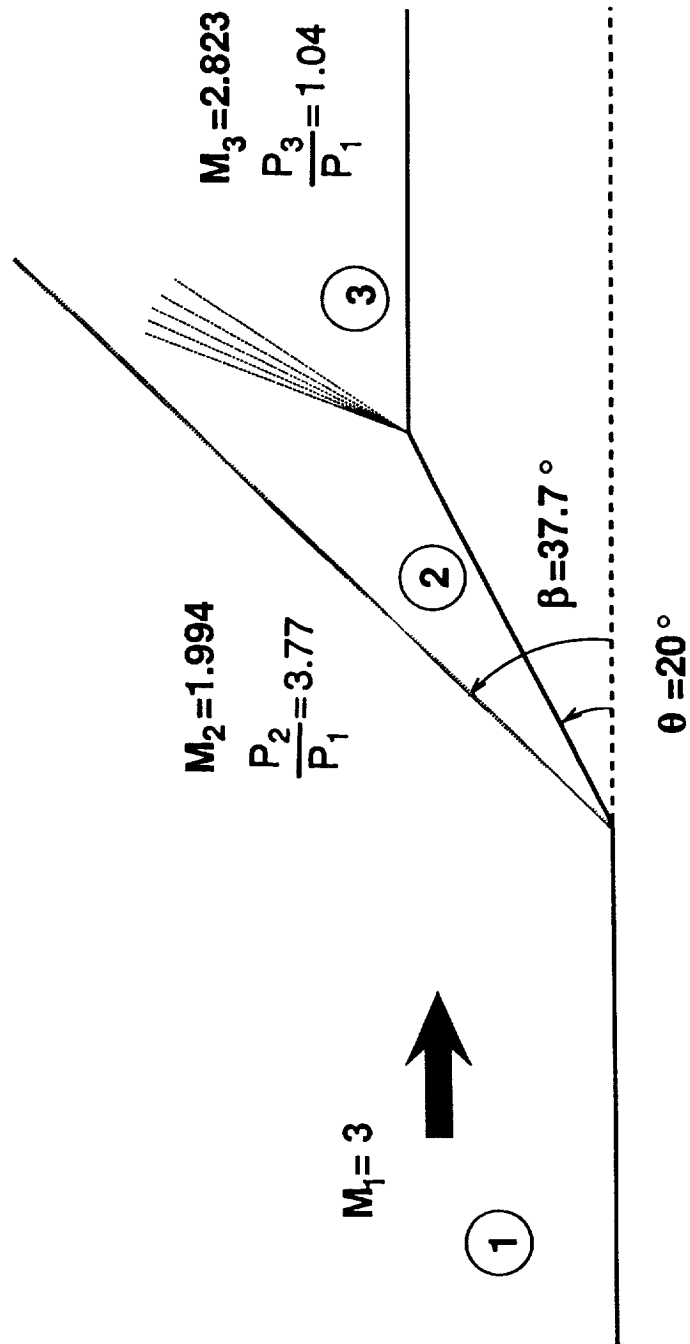# Pressure Coefficient Comparison for NACA 0012 Airfoil

The pressure coefficient distributions are compared in the figure. The results of the calculation made on the structured grid are designated by diamonds connected by straight lines. The results of the calculation on the unstructured grid are designated by filled circles. The solutions are very similar. The location and strength of the normal shock on the upper surface are nearly identical. The finite element solver captured the shock more sharply than did FLO52. The finite element solution shown was obtained in 4000 steps which used 1.9×10^{-3} seconds/point/timestep on a Convex C2 computer.

Comparison of NACA 0012 airfoil
at $M_\infty = .80$, $\alpha = 1.25°$

C_p

x/c

412
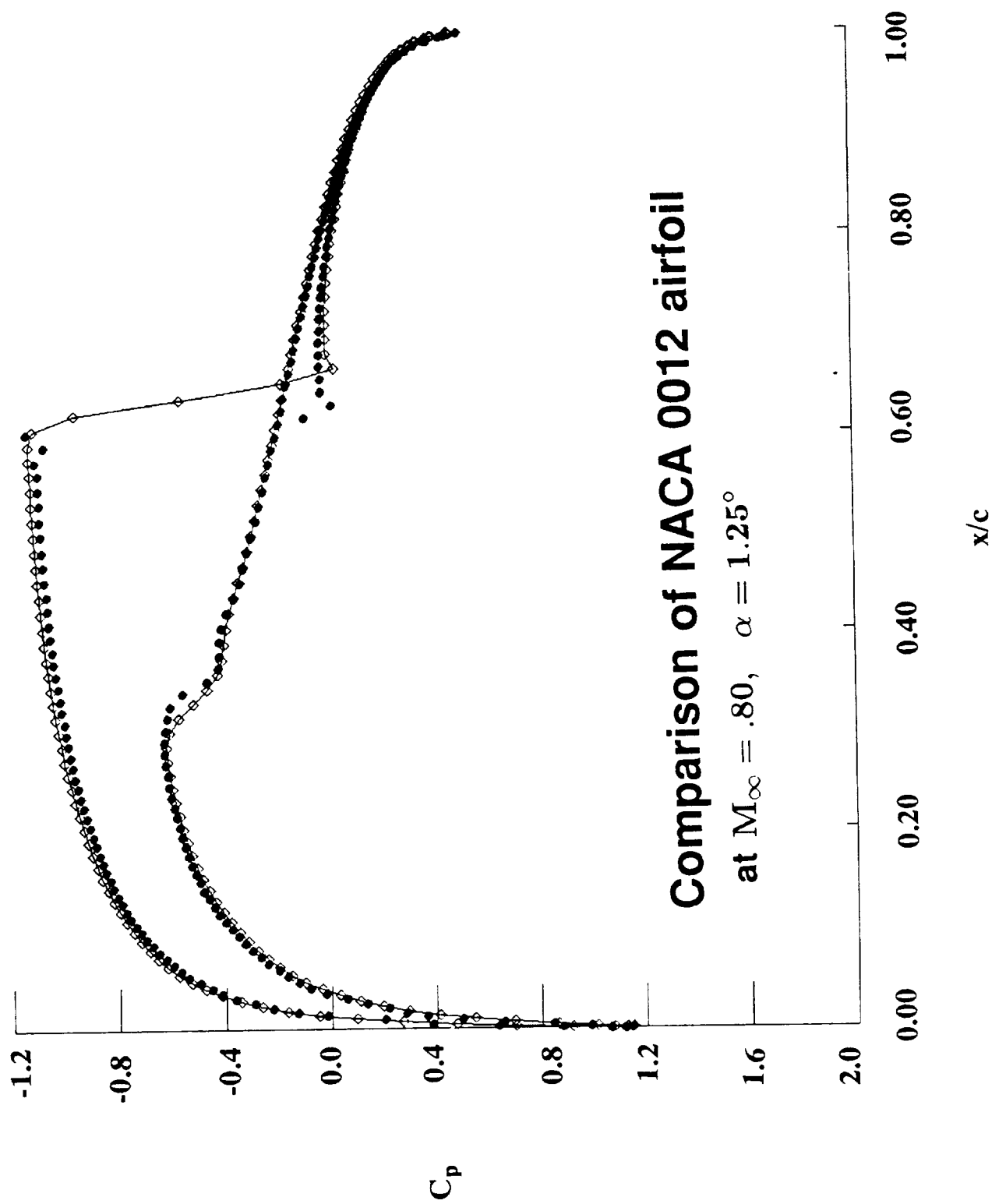
# Exact Solution For Supersonic Flow Over A Ramp

This figure shows a schematic of the second case, supersonic flow over a 20° wedge. The exact solution is shown for the regions designated 1, 2, and 3.

EXACT SOLUTION FOR SUPERSONIC FLOW OVER RAMP

$M_1 = 3$

①

$M_2 = 1.994$

$\dfrac{P_2}{P_1} = 3.77$

②

$\beta = 37.7°$

$\theta = 20°$

$M_3 = 2.823$

$\dfrac{P_3}{P_1} = 1.04$

③

414

Grid Adaptation for 20° Ramp

This figure shows the final adapted grid for the solution. The levels of adaptation are shown by different colors.
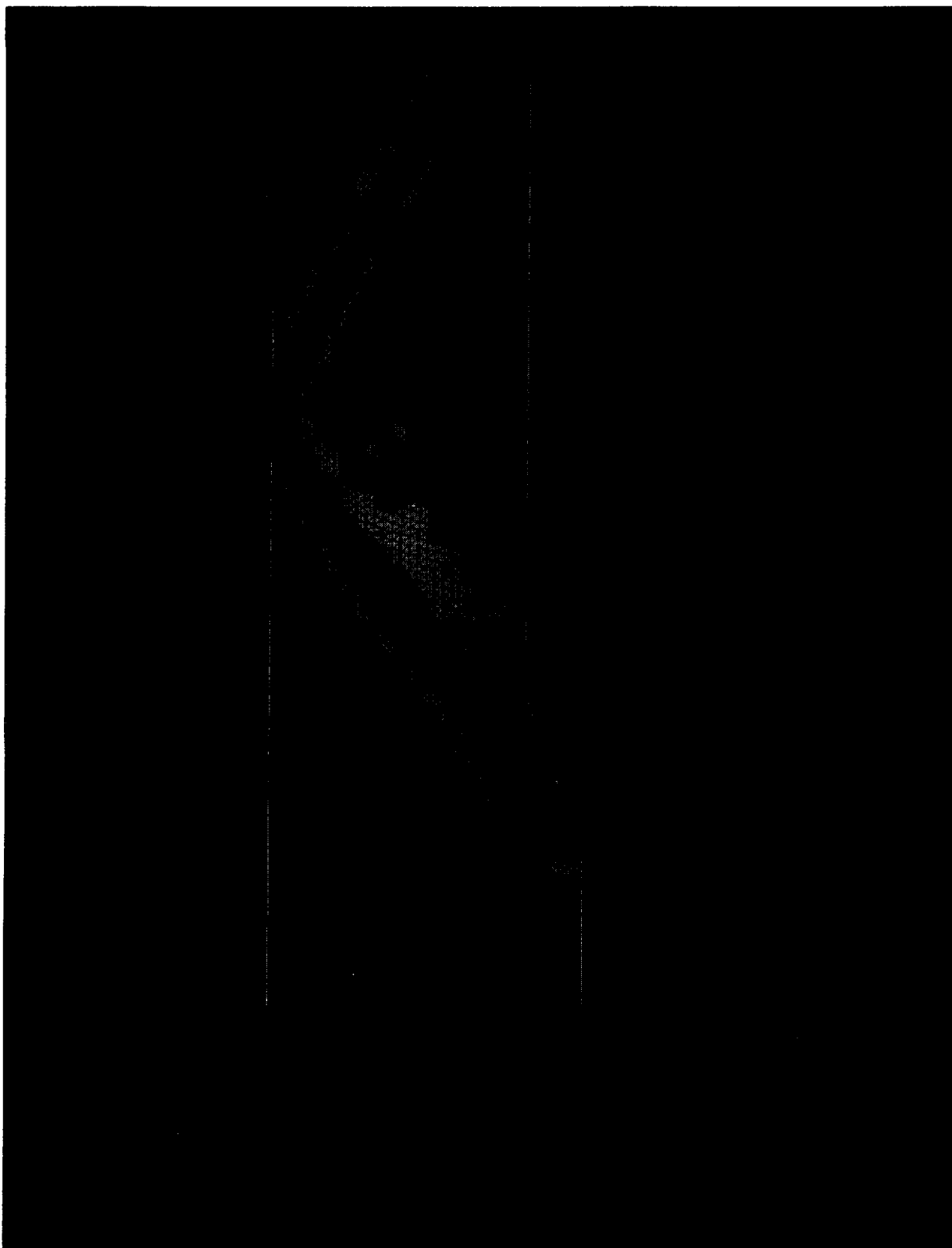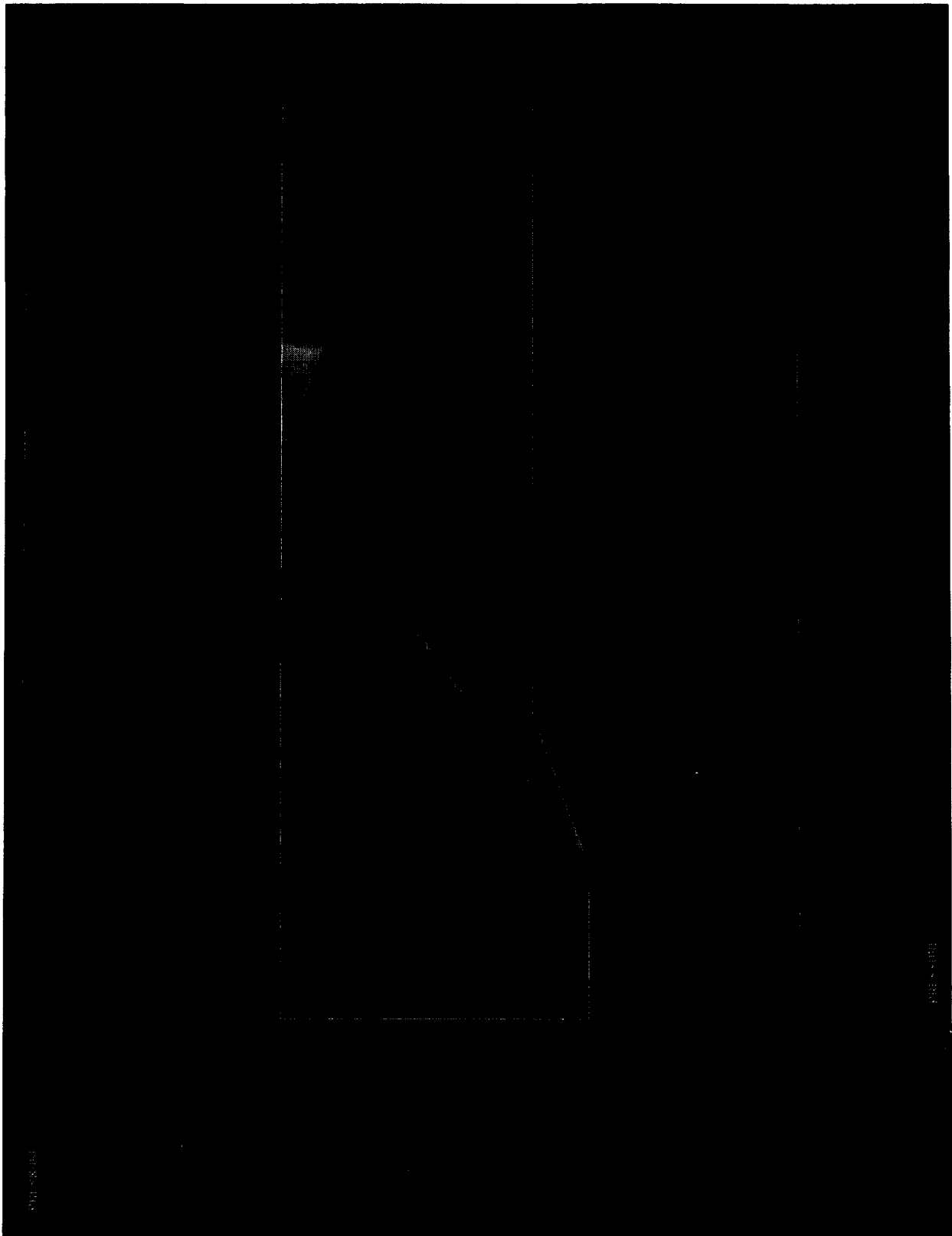
Mach Number and Pressure Contours for 20° Ramp

The calculated Mach number and pressure are shown in the following figures. The shock angle and the conditions in the regions designated 1, 2, and 3 are predicted accurately. The figure also shows the interaction of the oblique shock with the expansion fan and the reflection of the shock off the straight wall.

Surface Pressure Contours on F-18 Aircraft

The figure shows the pressure contours on the surface of an F-18 aircraft configuration at $M_\infty = 0.3$ and $\alpha = 15°$. The solution was calculated on a grid for which the engine inlet and nozzle were blocked. This solution was run 600 iterations in 159 minutes on the NAS CRAY-2 computer to decrease the maximum residual by two orders of magnitude.
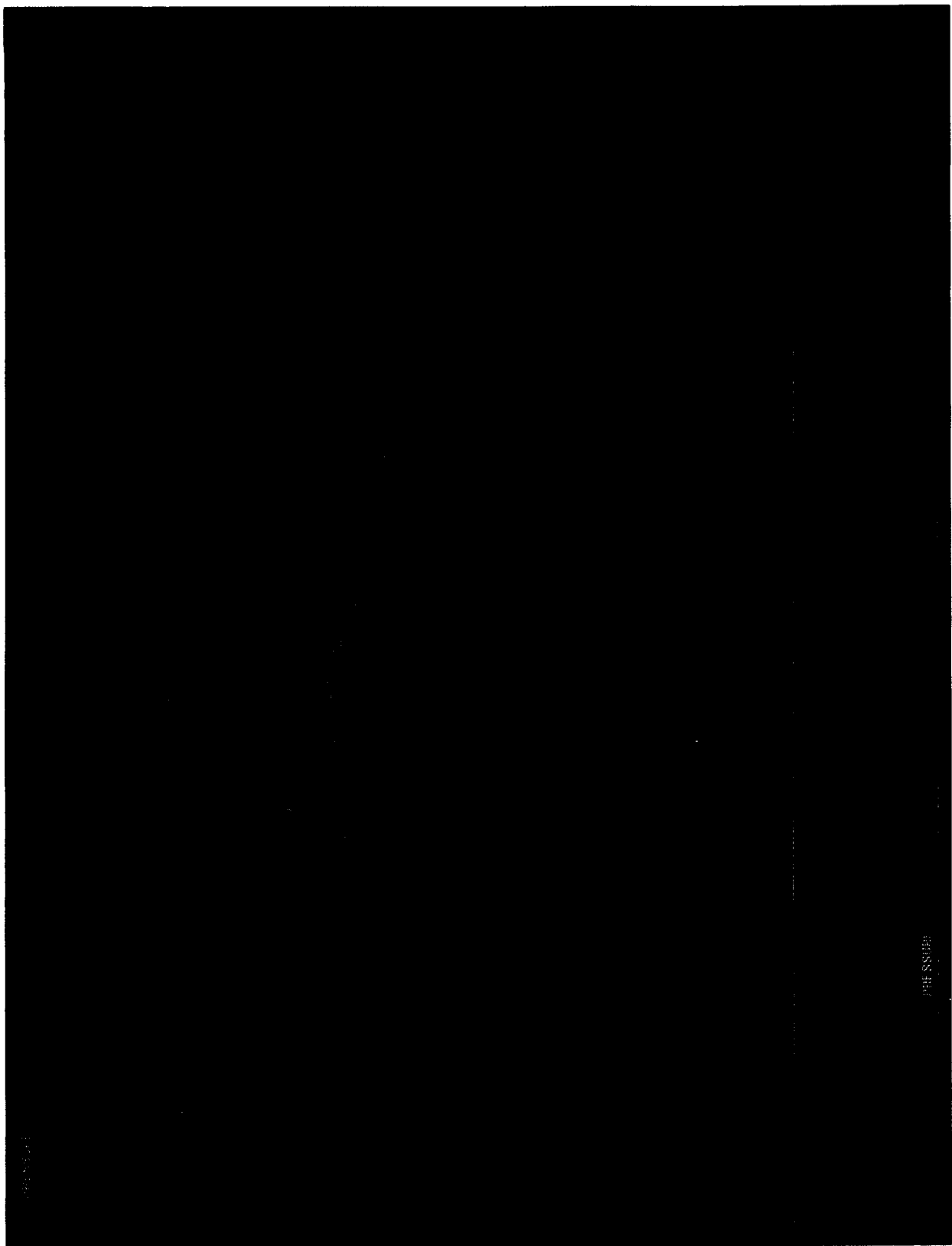
Pressure Contours For Shock Emanating From A Corner

Sharp gradients in density and pressure were imposed at the intersection of three perpendicular walls as the initial conditions for a transient flow calculation. The figures show the pressure contours on the walls at two different time steps. The grid on the three walls is also shown. Additional grid points were added during the calculation to better capture the sharp gradients. No derefinement was done after the large gradients had passed.
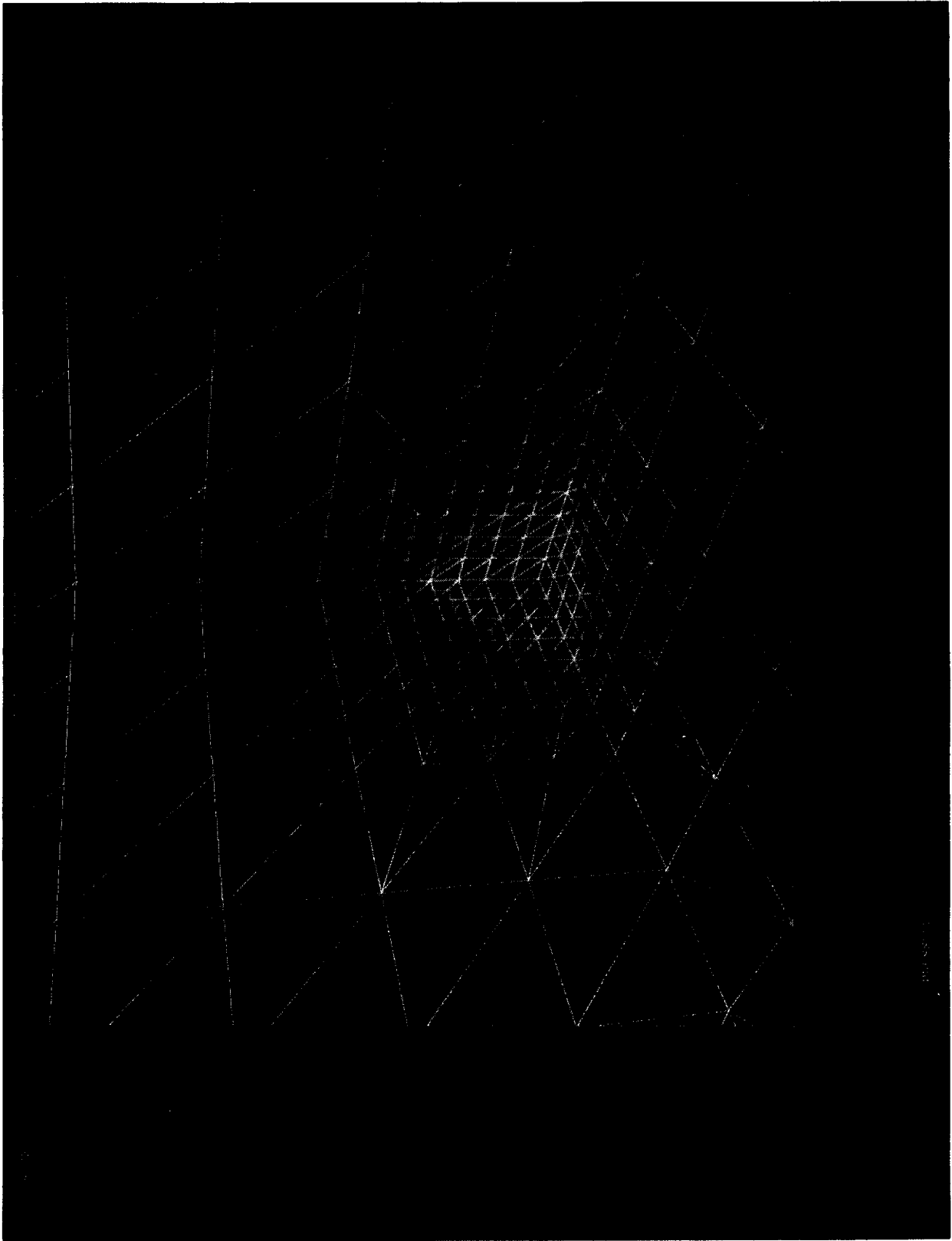
Future Plans

Further development of the grid generator will tend along three paths: (1) implementation of existing CAD sytem for faster and easier surface definition, (2) enhancement of the surface element library to allow slope continuity across surface patch boundaries and (3) faster and better ways of defining the background grid.

Further development of the flow solver is expected to involve incorporation of the viscous terms into the flow equations, use of implicit or semi-implicit algoriths and implementation on parallel computers.

FUTURE PLANS

- For the grid generator:

  - Link to commercial CAD/CAM systems for faster and more accurate surface definition.

  - Faster and better ways to input background grid information.

  - Enhancement of surface element library.

- For the flow solver:

  - Incorporation of viscous terms.

  - Implicit or semi-implicit flow solver for transient problems.

  - Implementation on parallel computers.

436