DEVELOPMENT OF A GRIDLESS CFD METHOD

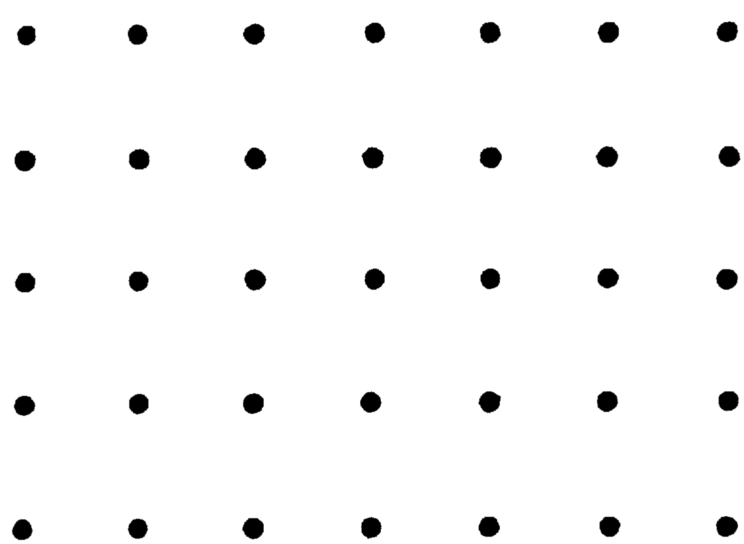
JOHN T. BATINA
NASA LANGLEY RESEARCH CENTER

PRESENTATION OBJECTIVE

- Leave you with some thoughts or ideas on an alternative approach to discretizing fluid flow problems (namely the so-called gridless approach)
- Ask you today to:
  - Expand your thinking
  - Be unconventional
- Why? Because if you expand the possibilities for generating grids or developing solution algorithms you might actually discover techniques that are superior to conventional procedures!

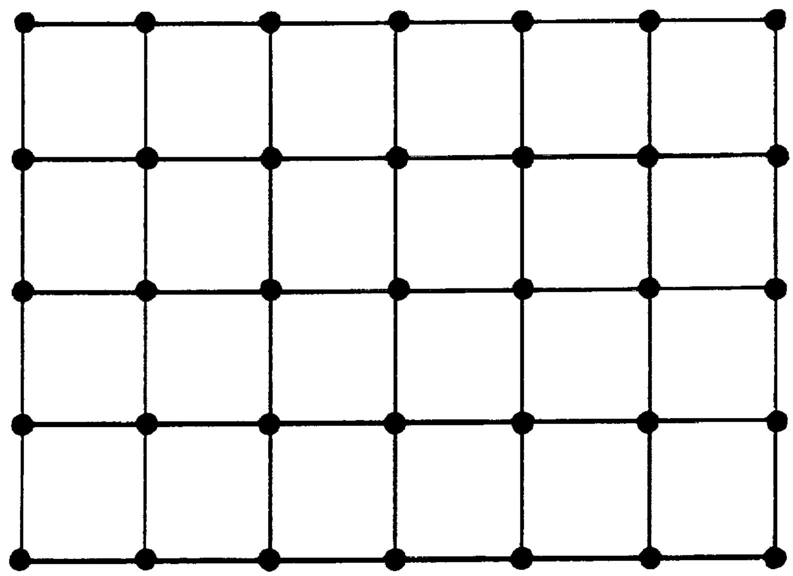
CONSIDER A SET OF POINTS IN A TWO-DIMENSIONAL DOMAIN

- How do you connect the points?



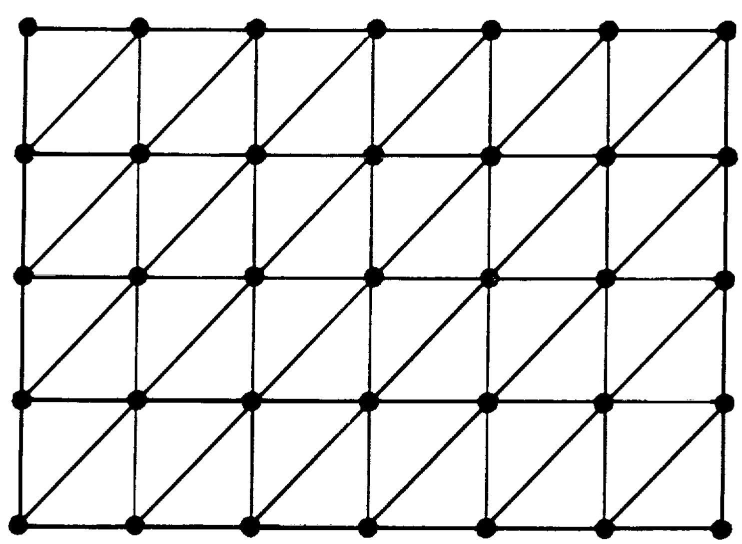
STRUCTURED GRID

- Should the points be connected in a structured fashion?



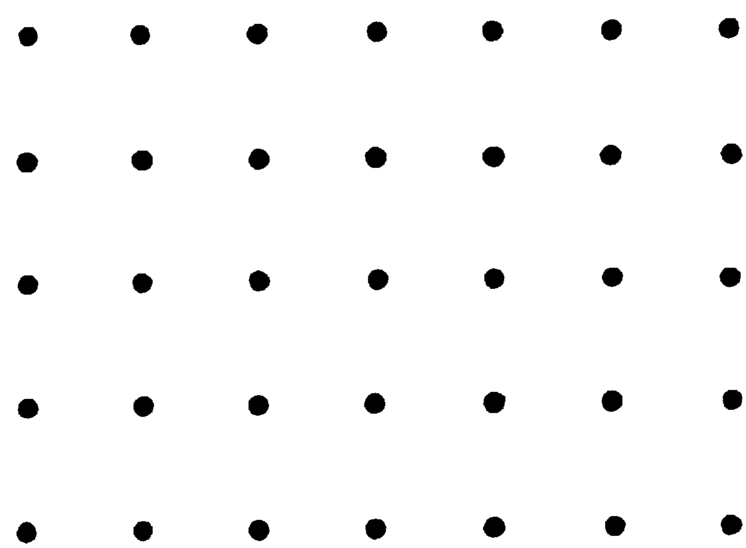
UNSTRUCTURED GRID

- Or should they be connected as an unstructured grid of triangles?



FIELD OF POINTS

- Maybe the points didn't need to be connected in the first place!



MOTIVATION FOR ALTERNATIVE APPROACH

- Tetrahedral meshes have an excessively large number of cells than structured grids
- These meshes, while reasonably adequate in the streamwise direction, tend to be much finer in the spanwise direction than is necessary for accurate flow computation
- Furthermore, for viscous applications, the additional requirement that the mesh be fine near the body, exacerbates the inefficiency
- The basic problem is that the tetrahedron is an inefficient geometrical shape

INTRODUCTION OF GRIDLESS APPROACH

- To alleviate the problem, some researchers have put structure back into the mesh in one coordinate direction
- This helps, but rather than take a step back toward grid structure, can we take a step forward and develop algorithms that do not require that the points be connected at all?
- This type of approach, referred to as "gridless," uses only clouds of points and does not require that the points be connected to form a grid as is necessary in conventional CFD algorithms
- The governing equations are solved directly, by performing local least-squares curve fits in each cloud of points, and then analytically differentiating the resulting curve fits to approximate the derivatives

SPATIAL DISCRETIZATION – DERIVATIVES

- Fluxes assumed to vary locally as

$$f(x,y,z) = a_0 + a_1 x + a_2 y + a_3 z$$

- a_0, a_1, a_2, and a_3 determined from a least-squares curve fit resulting in

$$\begin{bmatrix} n & \Sigma x_i & \Sigma y_i & \Sigma z_i \\ \Sigma x_i & \Sigma x_i^2 & \Sigma x_i y_i & \Sigma x_i z_i \\ \Sigma y_i & \Sigma x_i y_i & \Sigma y_i^2 & \Sigma y_i z_i \\ \Sigma z_i & \Sigma x_i z_i & \Sigma y_i z_i & \Sigma z_i^2 \end{bmatrix} \begin{Bmatrix} a_0 \\ a_1 \\ a_2 \\ a_3 \end{Bmatrix} = \begin{Bmatrix} \Sigma f_i \\ \Sigma x_i f_i \\ \Sigma y_i f_i \\ \Sigma z_i f_i \end{Bmatrix}$$

  where n is the number of points in the cloud and the summations are taken over the n points

- The spatial derivatives are now known since

$$\frac{\partial f}{\partial x} = a_1 \qquad \frac{\partial f}{\partial y} = a_2 \qquad \frac{\partial f}{\partial z} = a_3$$

SOLUTION BY QR – DECOMPOSITION

- Least-squares equations are of the form

$$\left(A^T A\right) a = A^T f$$

but $\left(A^T A\right)$ may be ill conditioned

- Instead the equations

$$Aa = f$$

are solved using a decomposition where $A = QR$ such that $Q^T Q = I$ and R is a square upper triangular matrix

- Solution given by

$$Ra = Q^T f$$

SPATIAL DISCRETIZATION – ARTIFICIAL DISSIPATION

- Artificial dissipation is added to the solution procedure since the method is conceptually analogous to a central-difference type approach

- Harmonic and biharmonic terms are added to the governing equations defined by

$$D = \nabla\left(\epsilon^{(2)}\lambda\right)\nabla Q - \nabla^2\left(\epsilon^{(4)}\lambda\right)\nabla^2 Q$$

where λ is the local maximum eigenvalue and $\epsilon^{(2)}$ and $\epsilon^{(4)}$ are local dissipation coefficients

- For the Navier-Stokes equations, an anisotropic model is used in part to account for the close spacing of points normal to the surface relative to the tangential distribution

BOUNDARY CONDITIONS

- Ghost points are used inside or outside of boundaries to impose the boundary conditions
- Along solid surfaces
  - velocity components determined by slip (Euler) or no-slip (Navier-Stokes) condition
  - pressure and density determined by extrapolation
- In the farfield
  - inviscid flow variables determined by a characteristic analysis based on Riemann invariants
  - viscous flow variables determined by extrapolation

TEMPORAL DISCRETIZATION – TIME INTEGRATION

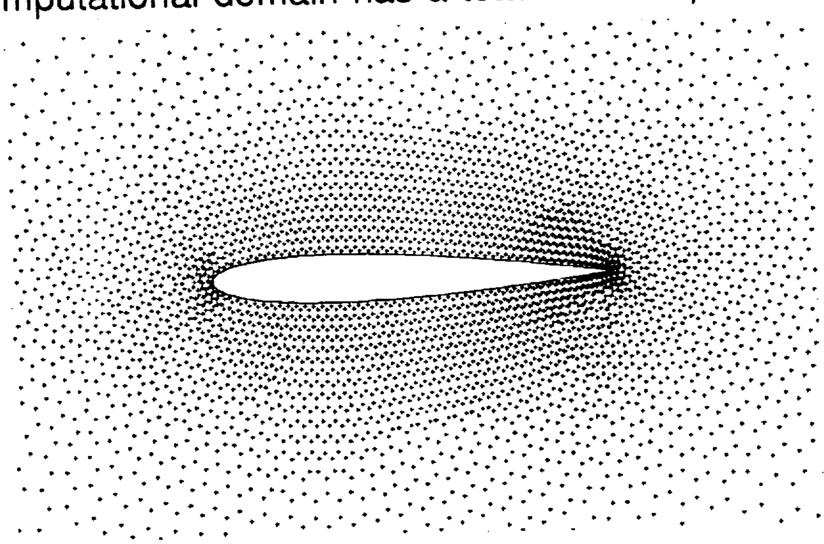
- Governing flow equations are integrated numerically in time using an explicit Runge-Kutta scheme
  - To solve the Euler equations, a four-stage scheme is used with the artificial dissipation evaluated only during the first stage
  - To solve the Navier-Stokes equations, a five-stage scheme is used with the artificial dissipation evaluated during the odd stages

OVERVIEW OF EULER RESULTS

- NACA 0012 airfoil
  - $M_\infty = 0.8$ and $\alpha = 0^\circ$
  - $M_\infty = 0.85$ and $\alpha = 1^\circ$
  - $M_\infty = 0.8$ and $\alpha = 1.25^\circ$
  - $M_\infty = 1.2$ and $\alpha = 7^\circ$
- ONERA M6 wing at $M_\infty = 0.84^\circ$ and $\alpha = 3.06^\circ$

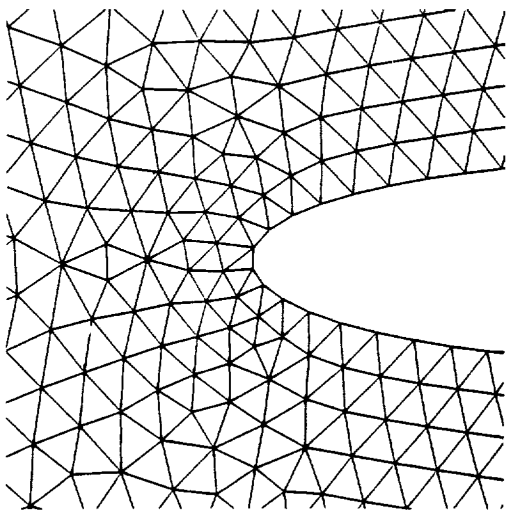
FIELD OF POINTS ABOUT NACA 0012 AIRFOIL

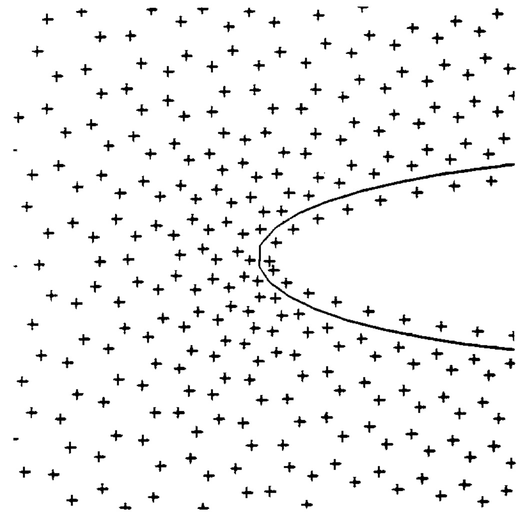
- Locations of points determined using the cell centers of an unstructured grid for convenience
- Computational domain has a total of 6500 points



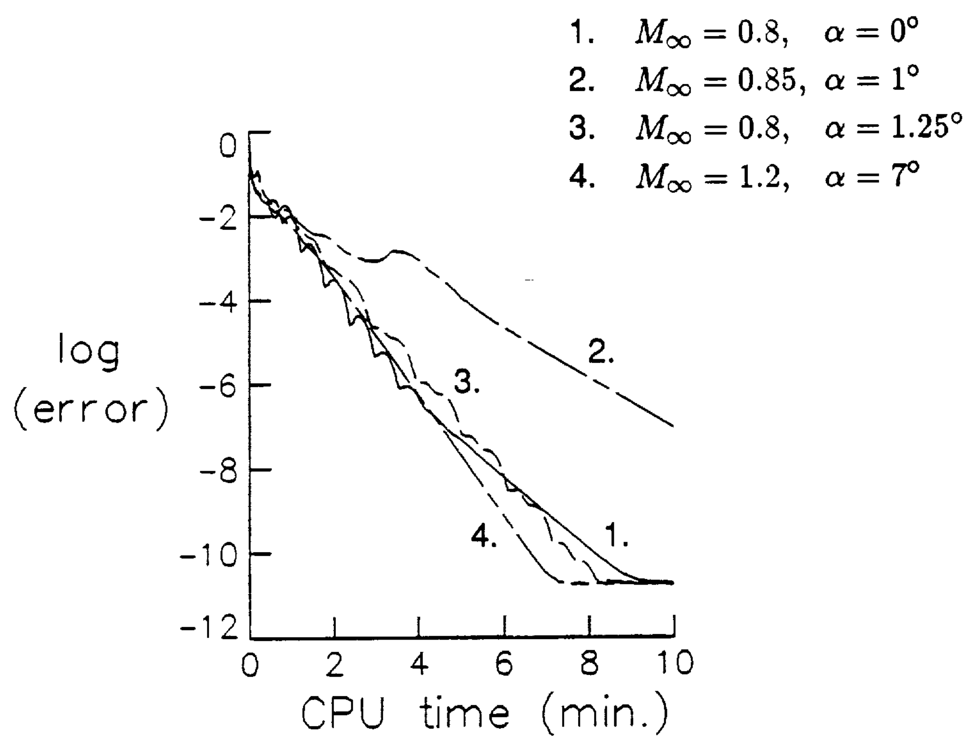
NEAR NOSE OF NACA 0012 AIRFOIL

- Unstructured mesh of triangles
- Corresponding field of points including ghost points

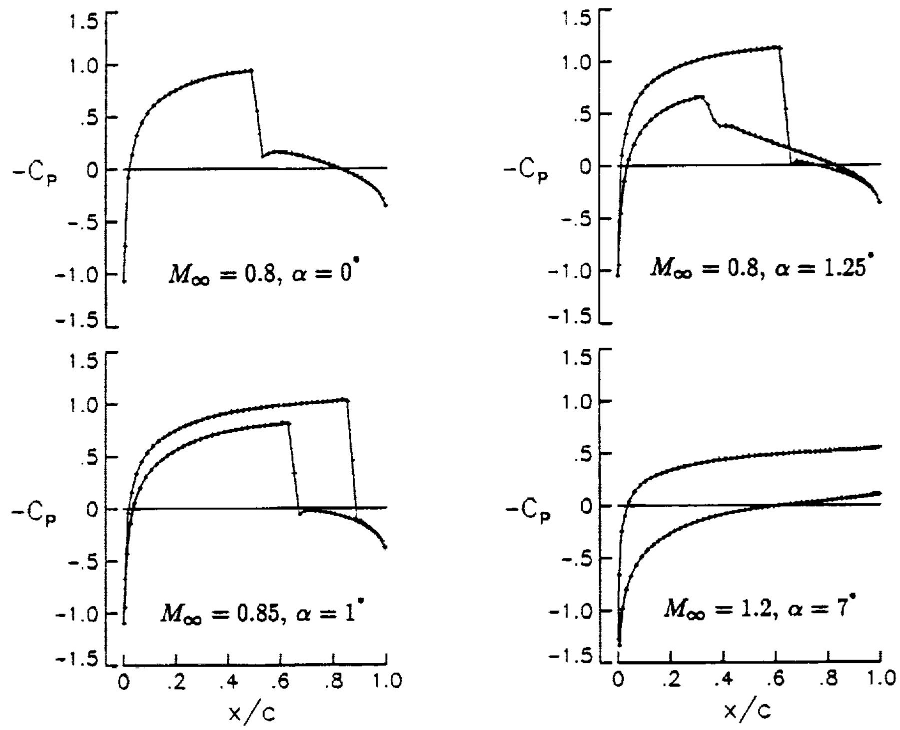




CONVERGENCE HISTORIES FOR NACA 0012 AIRFOIL

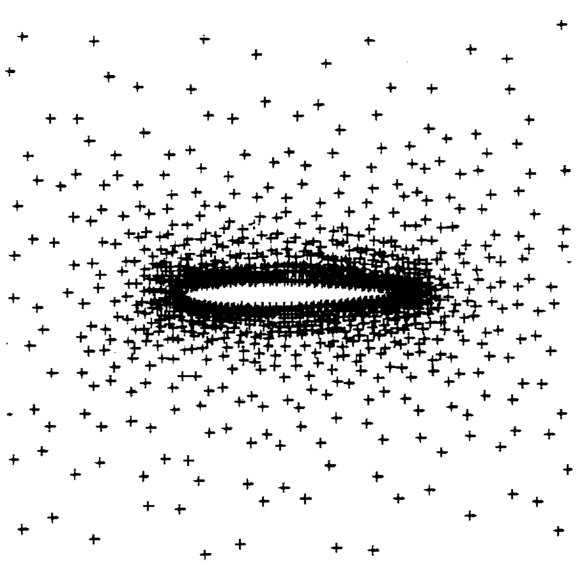


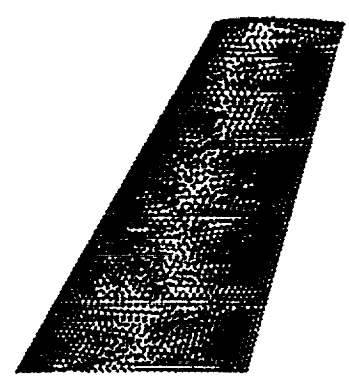
PRESSURE DISTRIBUTIONS FOR NACA 0012 AIRFOIL



GHOST POINTS FOR ONERA M6 WING

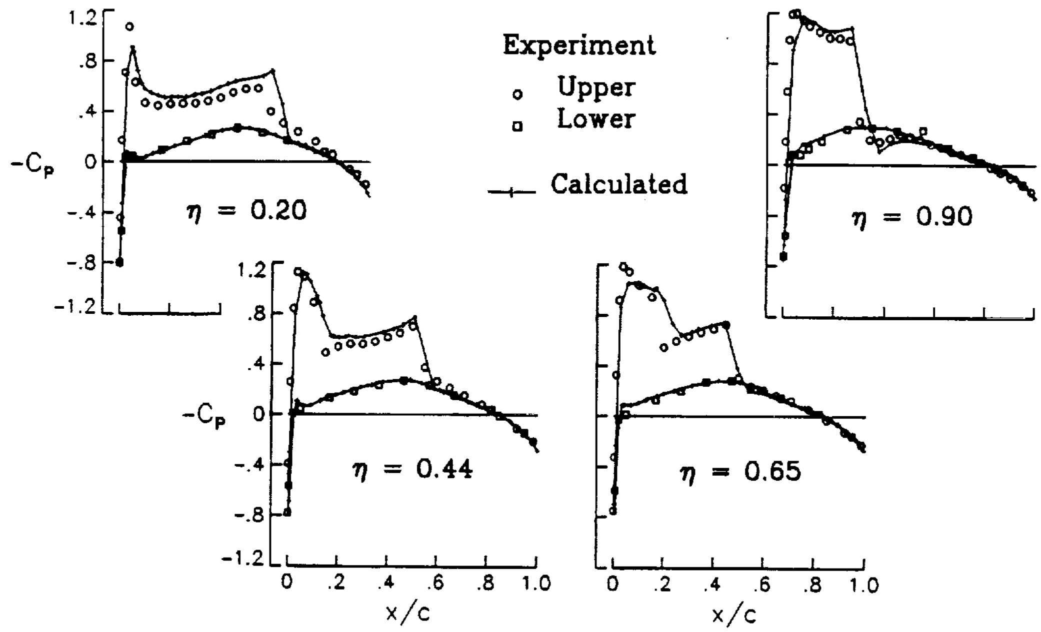
- Computational domain has a total of 108,705 points
- Symmetry plane
- Planform





EULER SOLUTION FOR ONERA M6 WING AT $M_\infty = 0.84$ AND $\alpha = 3.06°$

- Pressure coefficient distribution



ADVANTAGES/DISADVANTAGES OF GRIDLESS METHOD

- Gridless method is not faster on a per point basis in comparison with methods developed for structured or unstructured grids
- Advantage is that it allows the use of fields of points where the points are more appropriately located and clustered, leading to far fewer points to solve a given problem
- Method retains the advantages of the unstructured grid methods
  - general geometry treatment
  - spatial adaptation
- Disadvantage is that it requires indirect addressing to store cloud to point information

SUMMARY

- Development of a gridless method for the solution of the 2D and 3D Euler and Navier-Stokes equations was described
- Method uses only clouds of points and does not require that the points be connected to form a grid as is necessary in conventional CFD algorithms
- Calculations for standard Euler and Navier-Stokes cases were found to be reasonably accurate and efficient in comparison with alternative methods and experimental data

FINAL THOUGHTS

- The advent of gridless CFD does not obviate the need for "grid" generation — just the opposite
- Gridless CFD still requires surface definition and opens up the need to develop techniques for generating fields of points (in place of grids of points)