ORIGINAL CONTAINS
COLOR ILLUSTRATIONS

THE USE OF A NAVIER-STOKES CODE IN THE WING DESIGN PROCESS

S. Naomi McMillin
NASA Langley Research Center
Hampton, Virginia 23665-5225

Abstract of Paper Presented at the
NASA Computational Fluid Dynamics Conference
NASA Ames Research Center
March 6-9, 1989

An ongoing investigation is being conducted in the Supersonic/Hypersonic Aerodynamics Branch at NASA Langley Research Center to determine the feasibility of incorporating the Navier-Stokes computational code, CFL3D, into the supersonic wing design process. The approach taken in this investigation is of two steps.

The first step was to calibrate CFL3D against existing experimental data sets obtained on thin sharp-edged delta wings. The experimental data identified six flow types which are dependent on the similarity parameters of Mach number and angle of attack normal to the leading edge. The calibration showed CFL3D capable of simulating these various separated and attached-flow conditions.

The second step was to use CFL3D to study the initial formation of leading-edge separation over delta wings at supersonic speeds. This study consisted of examining solutions obtained on a 65^{o} delta wing at Mach number of 1.6 with varying cross-sectional shapes. Reynolds number was held constant at 1000000 and the Baldwin-Lomax turbulence model was used. The study showed that through the use of leading-edge radius and/or camber, the onset of leading-edge separation can be delayed to a higher angle of attack than observed on a flat sharp-edged wing.

Based on the geometries studied, three wind-tunnel models are being designed to verify these results. These models are to be tested over a Reynolds number range of 2×10^6/foot to 8.5×10^6/foot.

OBJECTIVE

- To determine the feasibility of incorporating the Navier - Stokes computational code, CFL3D, into the supersonic wing design process.

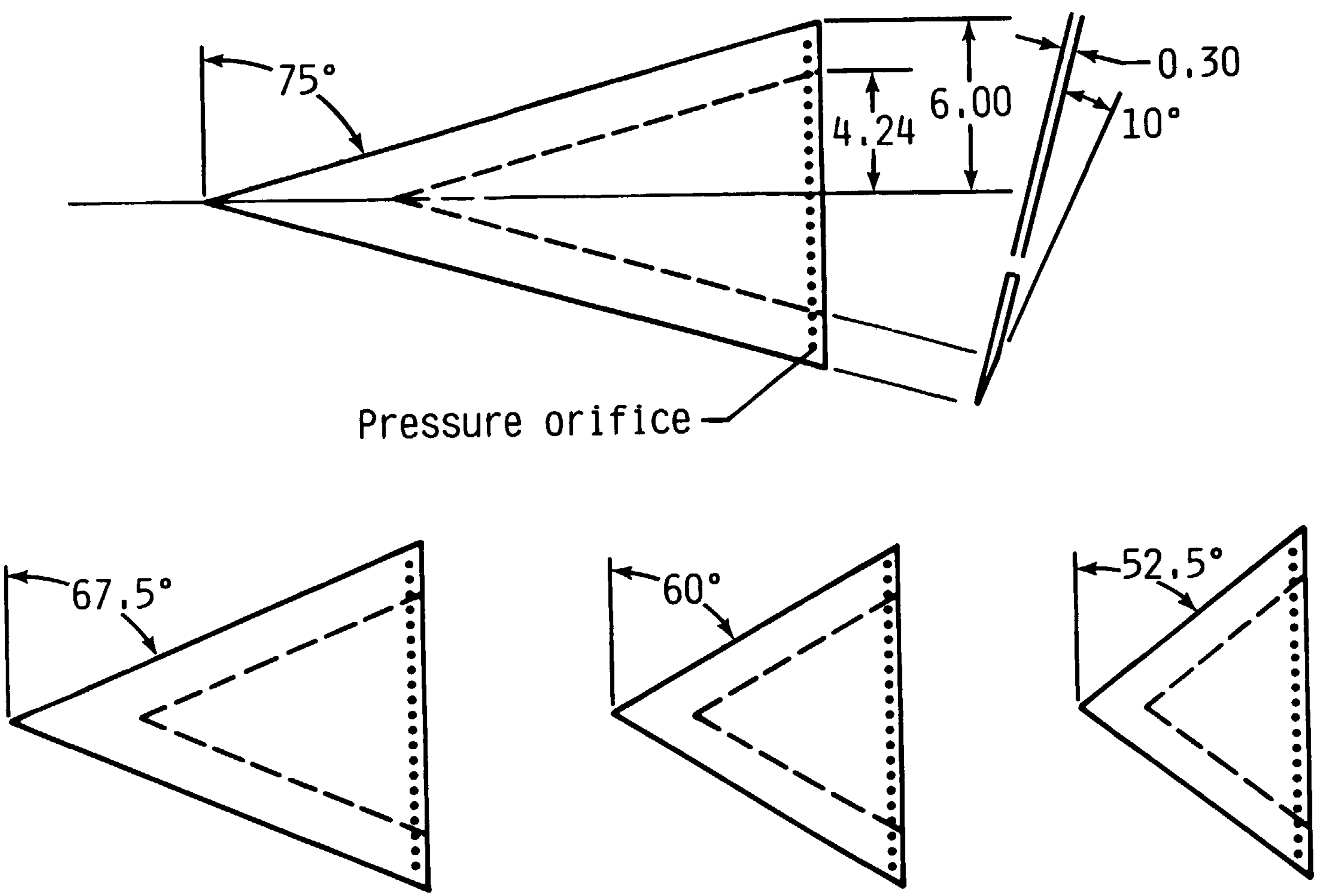
APPROACH

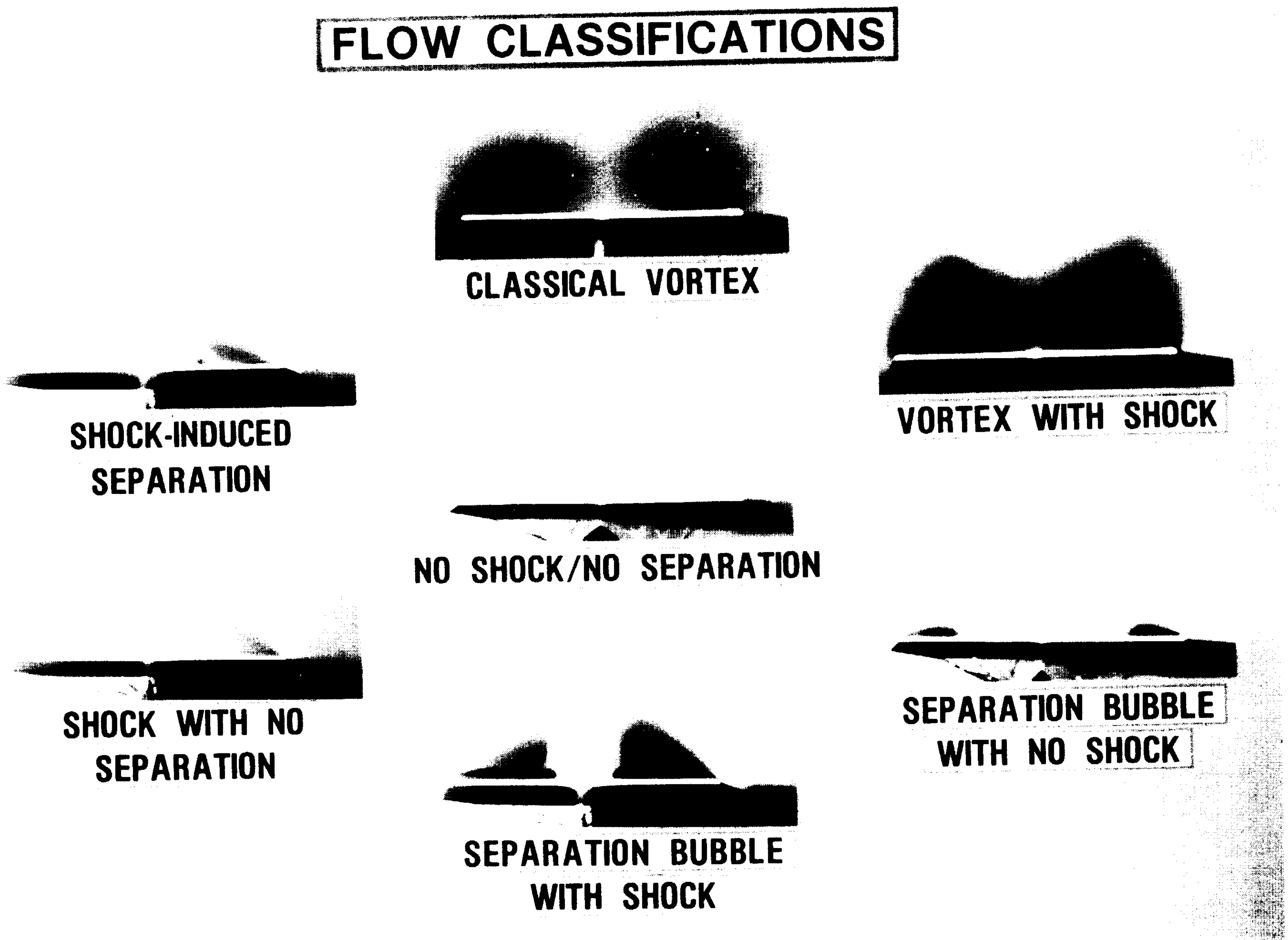
- Calibrate CFL3D with experimental data obtained on flat sharp-edged delta wings at supersonic speeds (AIAA 87-2270).
  Parameters:
  - leading edge sweep
  - angle of attack
  - Mach number
- Use CFL3D to study the initial formation of leading edge separation on delta wings.
  Geometric parameters:
  - leading edge radius
  - camber

CFL3D CODE CHARACTERISTICS

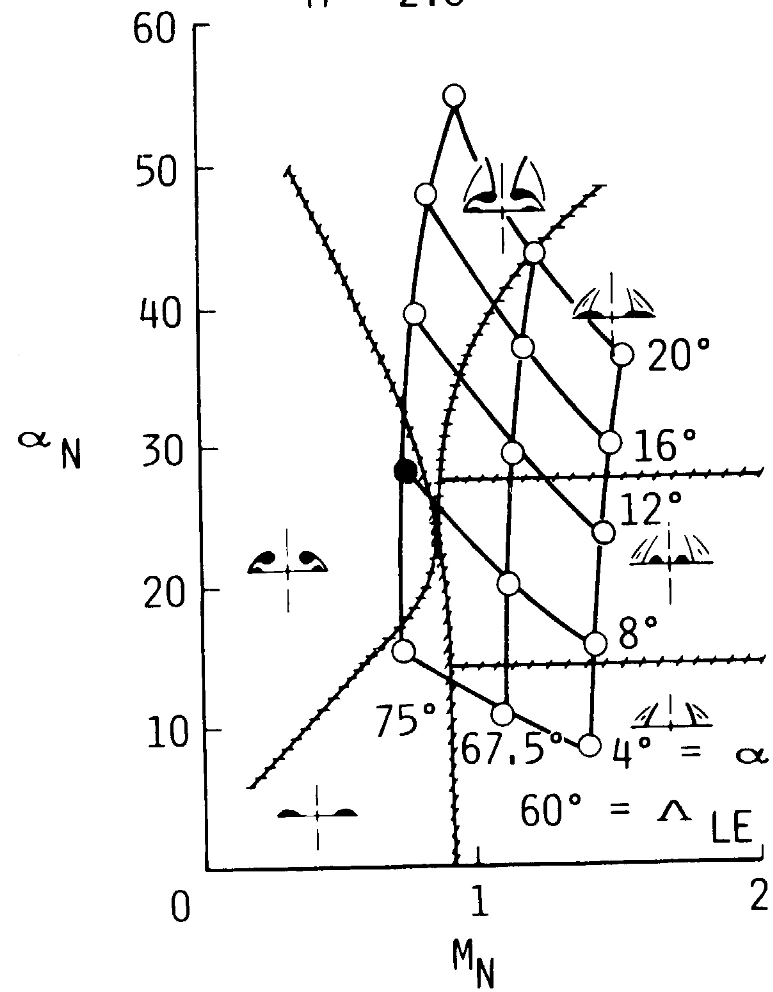
- Developed at NASA-LaRC by Jim Thomas
- Unsteady conservation-law form of compressible Navier-Stokes equations (thin layer laminar viscous model)
- Laminar viscous model or Baldwin-Lomax eddy viscosity turbulence model
- Upwind-biased spacial differencing implemented in finite volume form
    - Upwind (flux split) for convective terms and pressure
    - Central for shear stress/heat transfer
    - 2nd order accurate
- Implicit time differencing
    - Streamwise relaxation/cross-flow ADI
    - 1st order accurate
- Conical solutions obtained by 3-D solution
    - Single array of conically constructed volumes
    - Inflow equal outflow
- Grids generated using an elliptic grid generation method, 121 x 93 grid required to resolve details of flow patterns

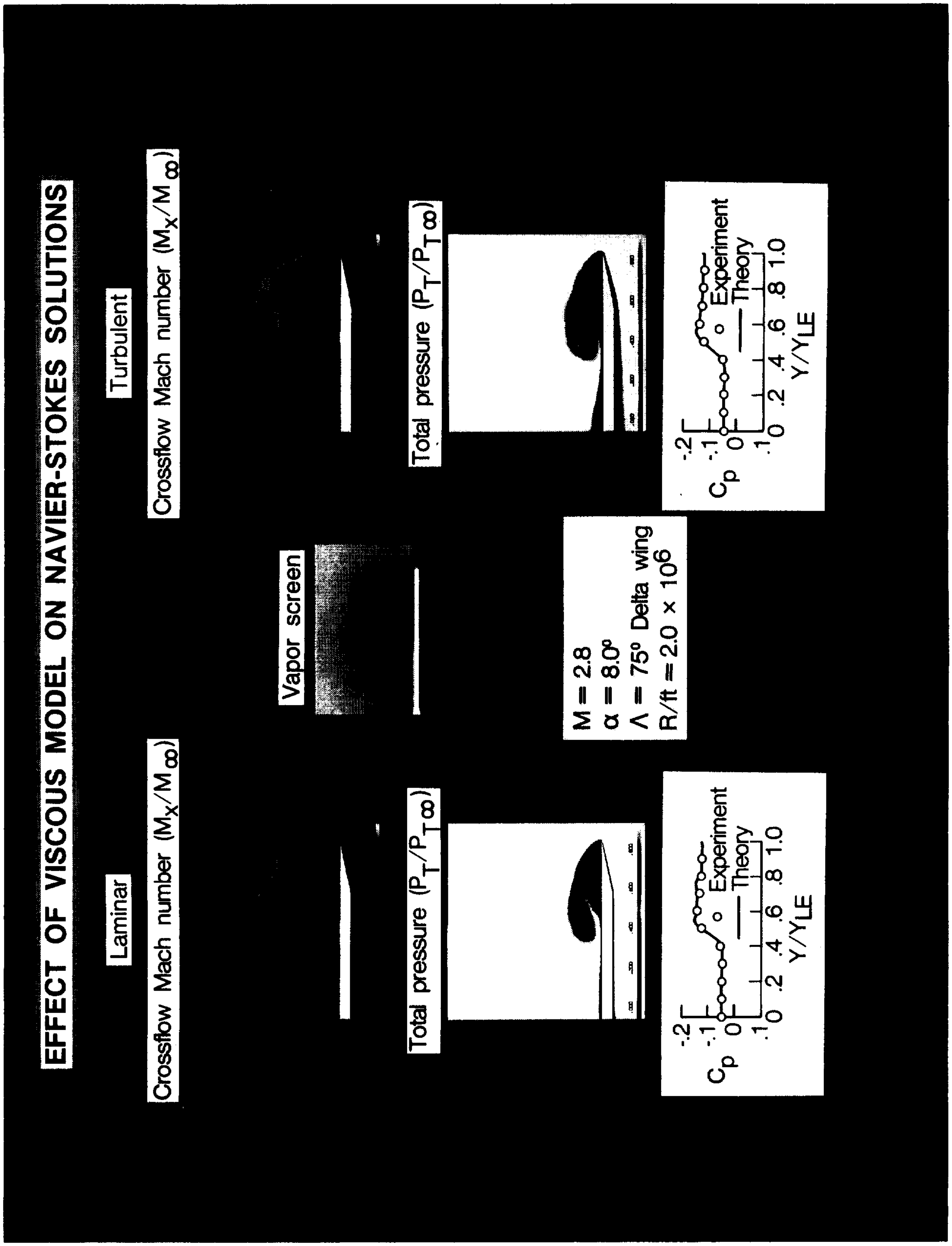
LEE-SIDE FLOW FLAT DELTA MODELS




FLOW CLASSIFICATIONS
CLASSICAL VORTEX
VORTEX WITH SHOCK
SHOCK-INDUCED SEPARATION
NO SHOCK/NO SEPARATION
SHOCK WITH NO SEPARATION
SEPARATION BUBBLE WITH NO SHOCK
SEPARATION BUBBLE WITH SHOCK

THEORETICAL TEST MATRIX

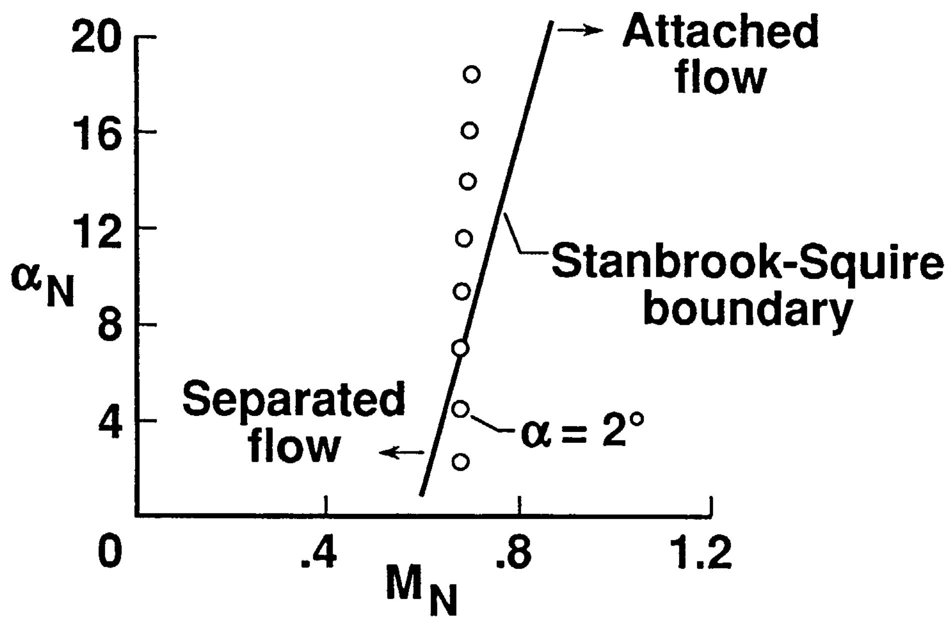




PRECEDING PAGE BLANK NOT FILMED

ORIGINAL PAGE IS OF POOR QUALITY

INCIPIENT SEPARATION STUDY



Examine effects of

- Leading edge radius
- Camber
- Angle of attack
- Reynolds number

On the formation of leading edge separation on a conical 65° delta wing at M = 1.6

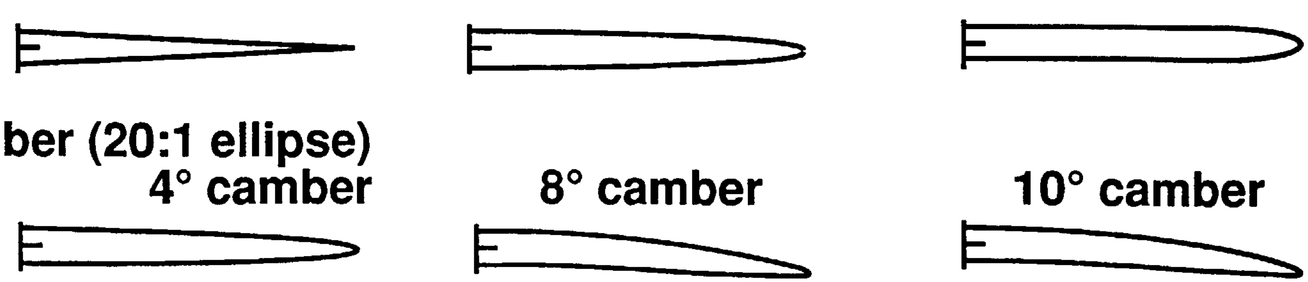
Parametric cross-sections

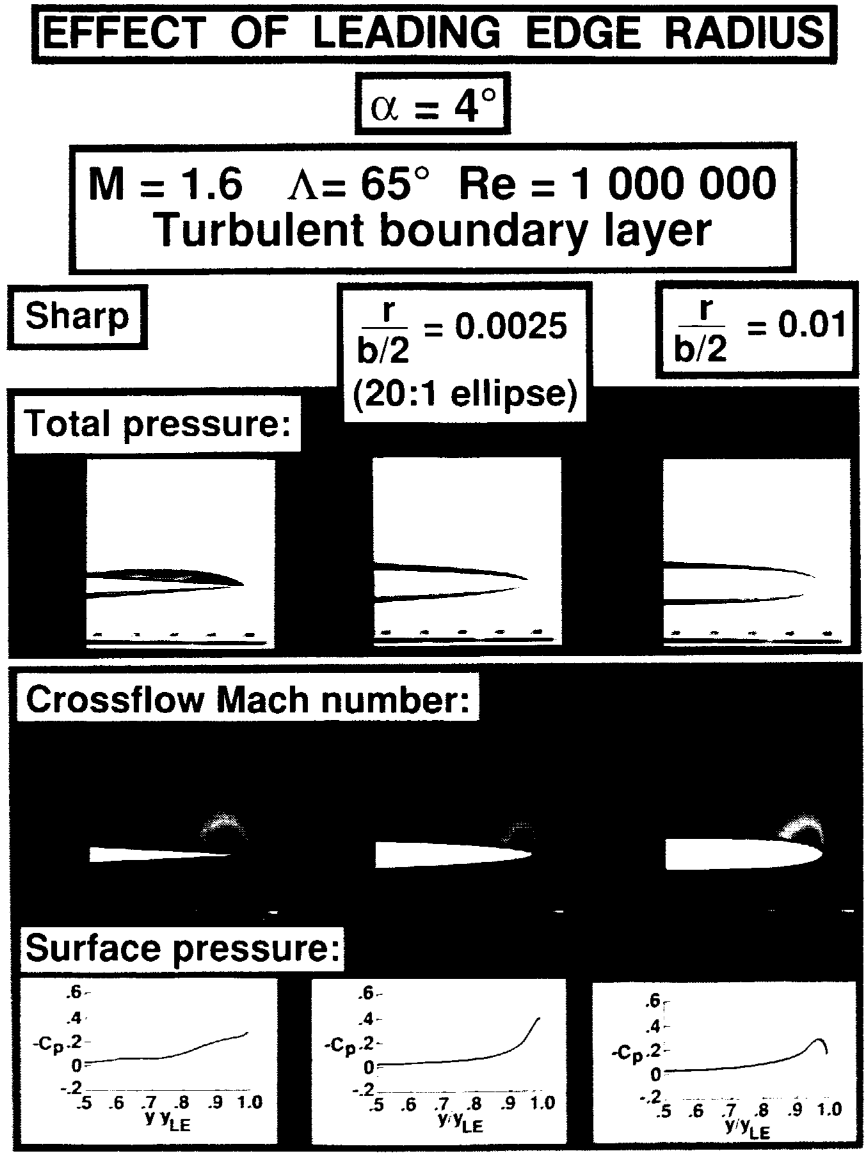
Leading edge radius:

Sharp | r/(b/2) = 0.0025 (20:1 ellipse) | r/(b/2) = 0.0100

Camber (20:1 ellipse)

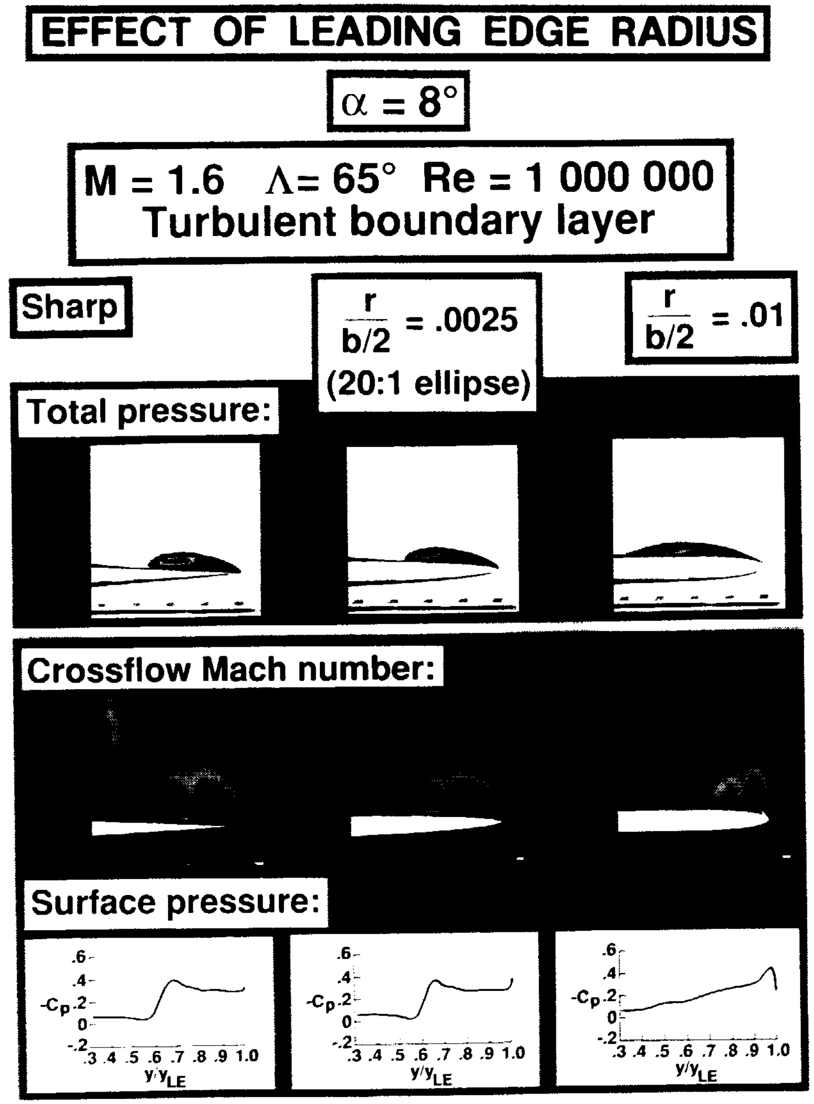
4° camber | 8° camber | 10° camber





PRECEDING PAGE BLANK NOT FILMED

ORIGINAL PAGE IS
OF POOR QUALITY



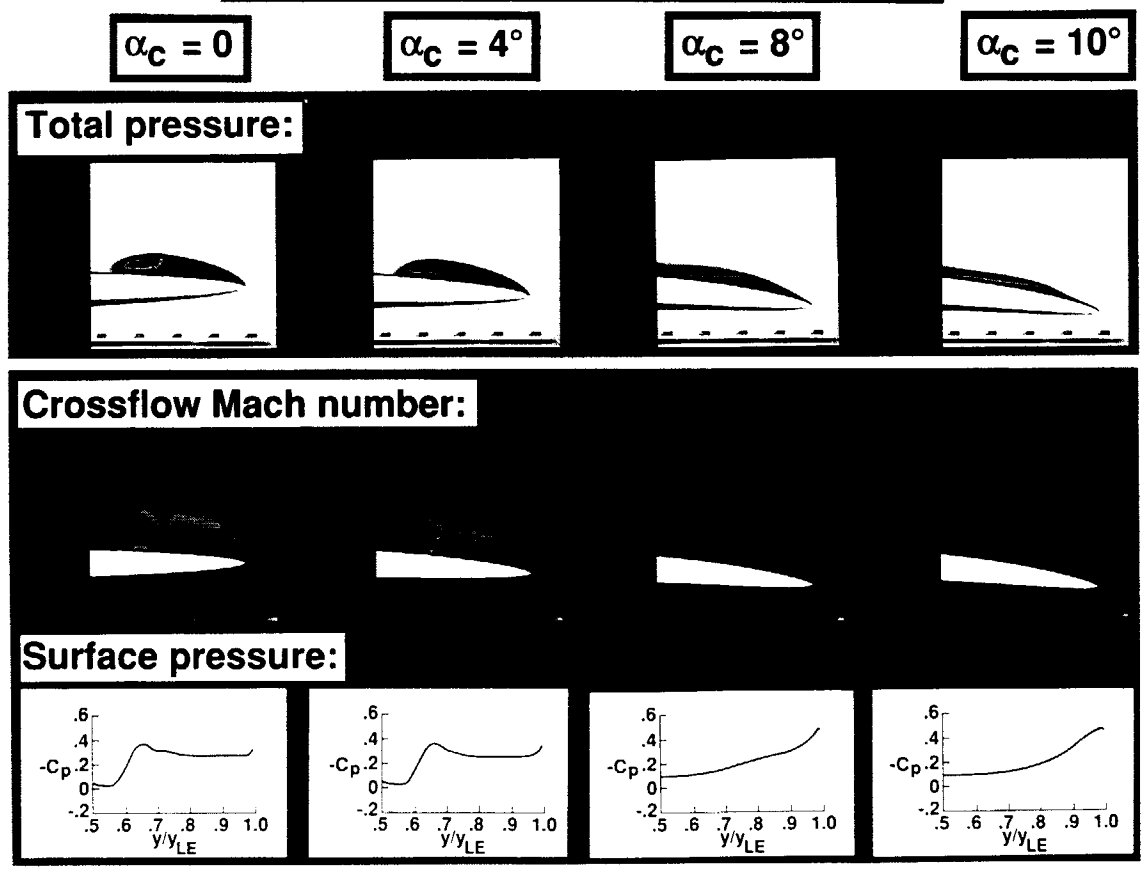
ORIGINAL PAGE IS
OF POOR QUALITY

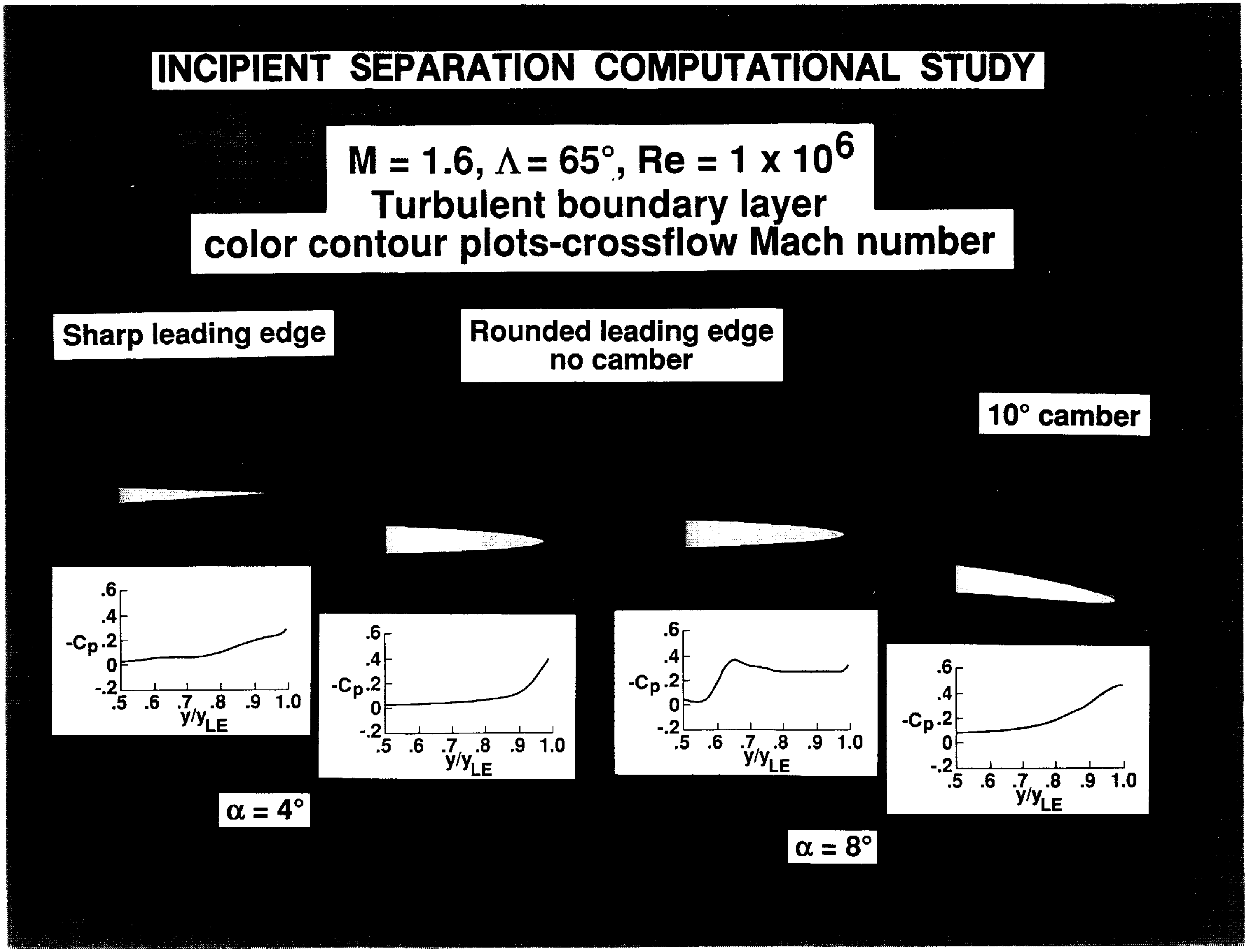
PRECEDING PAGE BLANK NOT FILMED

EFFECT OF CAMBER

$\alpha = 8°$

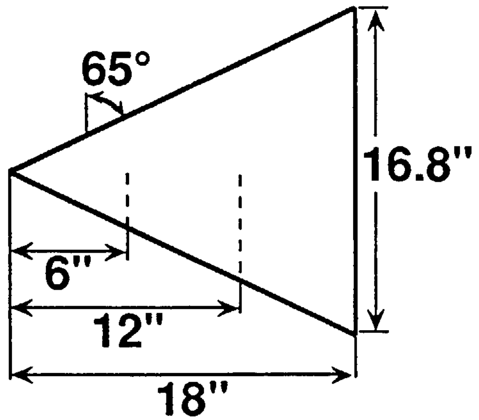
$M = 1.6 \quad \Lambda = 65° \quad Re = 1\,000\,000$
Turbulent boundary layer





PRECEDING PAGE BLANK NOT FILMED

WIND TUNNEL MODELS

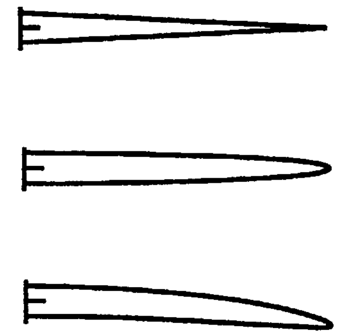


Cross-sections:

Sharp

20:1 ellipse

20:1 ellipse
with 10° camber



Wind Tunnel Test Description

- M = 1.6; α = 0° to 8° (initial conditions)
- Surface pressure data (~100 ports)
- Vapor screen photographs
- Oil flows
- Unitary Plan Wind Tunnel
  R e = 2×10^6 /foot, 5×10^6 /foot
- 20" Supersonic Wind Tunnel
  R e = 2×10^6 /foot, 8.5×10^6 /foot

CLOSING REMARKS

- The six distinct flow types observed over delta wings at supersonic speeds have been computationally reproduced using CFL3D (AIAA 87-2270)

- CFL3D has been used to quantify the effects of leading edge radius and camber on leading edge separation. Wind tunnel test designed to confirm these results.