

CR-200211

INTERIM  
IN-82-CR  
OCT  
8370  
p-31

Annual Progress Report  
Submitted to the Low Speed Aerodynamic Branch  
NASA Ames Research Center  
Moffett Field, CA

for

NASA Grant NAG2-905

entitled

"An Experimental Investigation of the Flow Physics  
of High-Lift Systems"

for the period

March - December 1995

by

Drs. Flint O. Thomas and R. C. Nelson  
Hessert Center for Aerospace Research  
Department of Aerospace and Mechanical Engineering  
University of Notre Dame  
Notre Dame, IN 46556

December 20, 1995

(NASA-CR-200211) AN EXPERIMENTAL  
INVESTIGATION OF THE FLOW PHYSICS  
OF HIGH-LIFT SYSTEMS Annual Report,  
Mar.-Dec. 1995 (Notre Dame Univ.)  
31 p

N96-19286

Unclas

G3/02 0101128

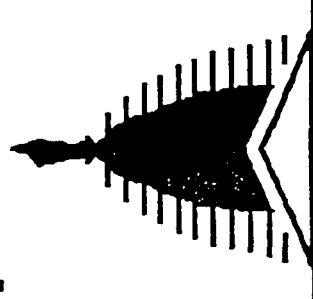
**A Progress Report:**

**Experiments on the Flow Physics of Confluent  
Boundary Layers for High Lift Systems**

**Supported by NASA Ames Research Center  
under NAG2-905**

**F. O. Thomas, R. C. Nelson and H. C. Chu**

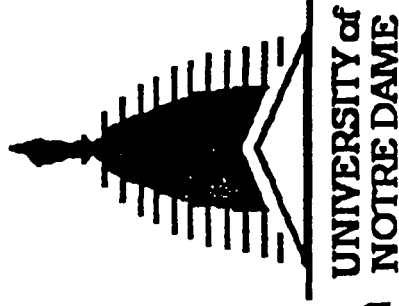
**Hessert Center for Aerospace Research  
Aerospace and Mechanical Engineering  
University of Notre Dame  
Notre Dame, IN**



**UNIVERSITY of  
NOTRE DAME**

# Organization

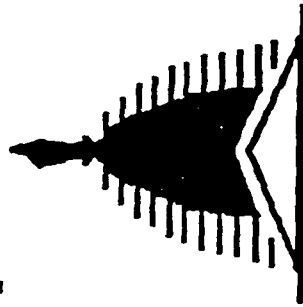
- The High-Lift Problem.
- Some Philosophy: “High-Lift Building Block Flows”
- Slat Wake / Main Element Confluent Boundary Layer
- Research Objectives
- Research Progress to Date
  - Project Time Line
  - Flow Visualization Study
  - Surface Pressure Measurements: Integrated Lift
  - LDV Survey
- Research Goals
- Questions/Comments/Discussion



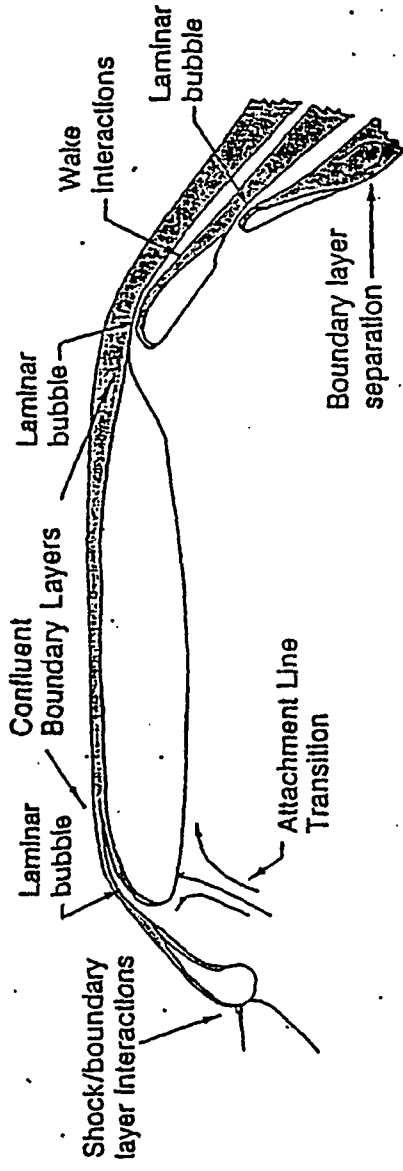
**Designer's Objective:** Design high lift systems with improved  $C_{L_{max}}$  for landing approach and improved take-off  $L/D$  and simultaneously reduce acquisition and maintenance costs. *In effect, achieve improved performance with simpler designs.*

**Example:** For a twin engine transport 1% increase in take-off  $L/D$  is equivalent to a 2800 lbf increase in payload or a 150 nautical mile increase in range (Meredith, 1993).

High lift systems pose a major constraint regarding sizing and overall configuration performance.



UNIVERSITY of  
NOTRE DAME



Viscous Interactions in High Lift Systems (Mack & McMasters, 1992)

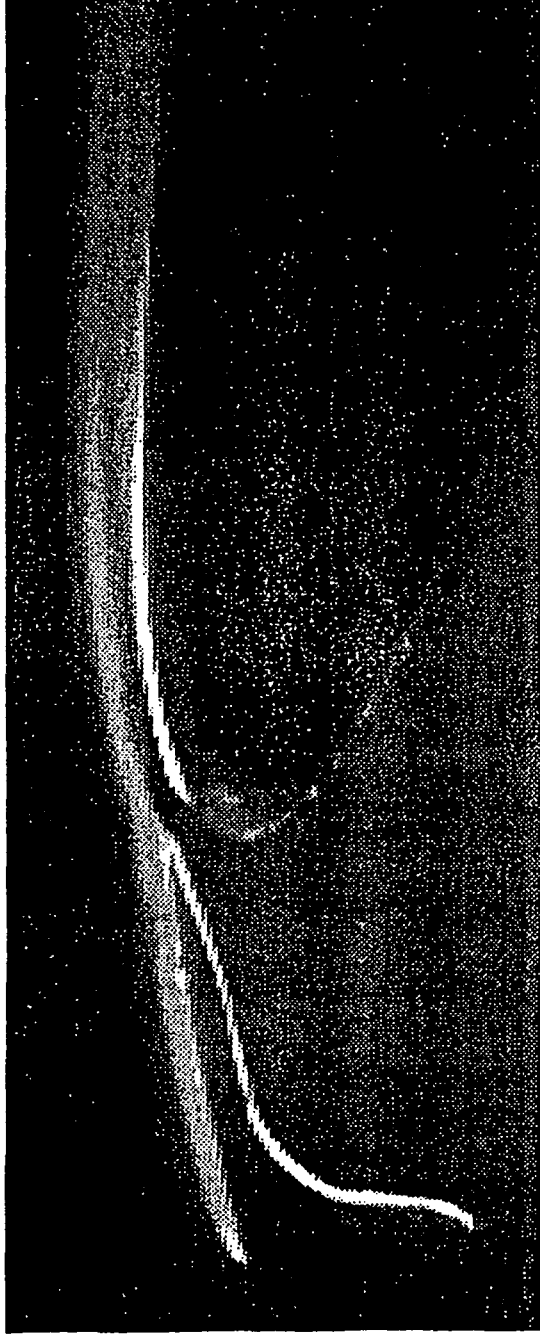
- Attachment Line Transition Mechanism
- Laminar Separation Bubbles
- Boundary Layer - Wake Interactions (the Confluent Boundary Layer)
- Wake Development in Pressure Gradient and with Streamline Curvature
- Multiple Wake Interactions
- Large-Scale Cove Flow Separation
- Turbulent Boundary Layer Development Under Influence of Pressure Gradient and Surface Curvature
- Turbulent Boundary Layer Relaminarization

- **Need for reliable CFD design tools.**
- **Much progress in grid generation methodology and algorithms**
- **Many codes represent two-dimensional flow calculations at fixed Reynolds number (e.g. AGARD High Lift Symposium, Banff, Alberta, Canada, 1992) . . . No Reynolds Number "dial."**
- **Transition mechanism not fully understood.**
- **Techniques not fully capable of accurately modeling complex viscous interactions, especially mixed flows.**
- **Model constants not known a priori for complex non-canonical flows encountered in high lift systems....must rely on experimental results.**
- **Computations are basically postdictive...depend on empirical inputs for mixing length and kinetic energy transport . . .Limits their utility as design tool.**

- High lift system flow field is dominated by complex viscous flow phenomena. As a result, design and development of high lift systems has traditionally occurred with heavy recourse to experimentation.
- Some aspects of the flow are inherently three-dimensional and may exhibit unsteady effects.
- There can be no doubt that improved design of high - lift systems is hindered by limited understanding of complex flow physics.
- The aerodynamic phenomena encountered are individually quite complicated and not fully understood. The best measure of understanding is predictive ability.
- In addition, they are coupled; this can lead to nonintuitive, complicated system performance.
- 2 - D pressure tunnel tests or semi-span tests at high Reynolds number provide important performance data but insight into the fundamental flow field physics issues may be obscured by the inherent complexity of the configuration.
- Need to individually understand high lift system “building block flows” before there can be any realistic hope of having a predictive capability for flow fields in actual high lift systems.

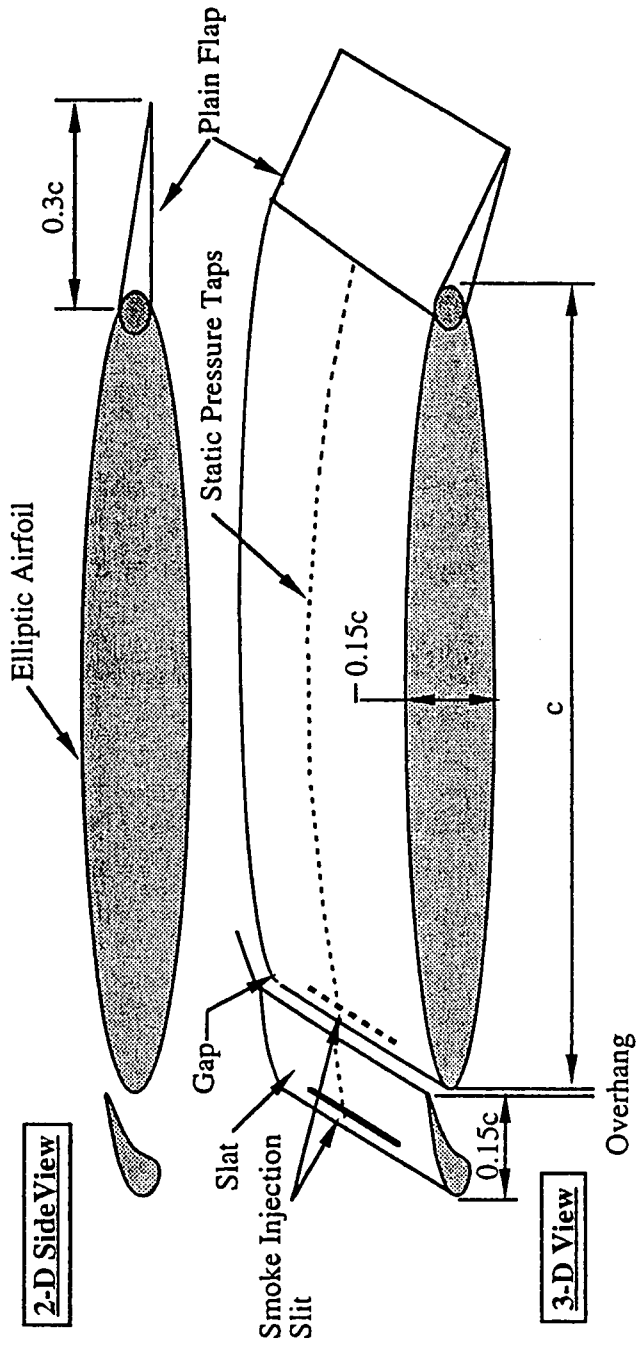
## High Lift System Building Block Flows: Confluent Boundary Layer

- Refers to Wake / Boundary Layer Interaction

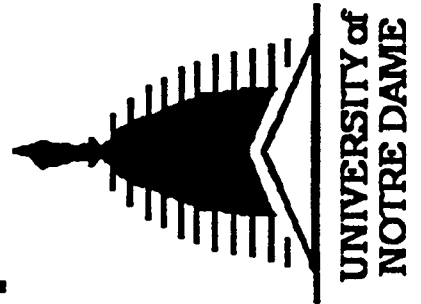


Kerosene smoke injection into slat wake and airfoil boundary layer visualized with Argon ion laser light sheet  
Hessert Center for Aerospace Research, University of Notre Dame





Schematic of High-Lift System Model



# *Research Objectives*

---

- **Establish the Role of Confluent Boundary Layer Flow Physics in High-Lift Production.**
- **Contrast Confluent Boundary Layer Structure for Optimum and Non-optimum  $C_L$  Cases.**
- **Formation of a High Quality, Detailed Archival Data Base for CFD / modelling.**
- **Examination of the Role of Relaminarization and Streamline Curvature.**

Schedule

Project Phase and Description

Months from start of grant

0    4    8    12    16    20    24    28    32    36

Phase 1 Slat/Main Wing Confluence Study

Model Design and Construction

Design

Construction



Flow Visualization

Develop flow visualization techniques for measuring location of confluence.



Determine cases of strong and weak confluence.



Detailed Flow Surveys

Select cases of strong and weak confluence for detailed LDA surveys

Validation



Pressure Distribution Study

Determine pressure distribution over main wing for selected cases of strong and weak confluence.



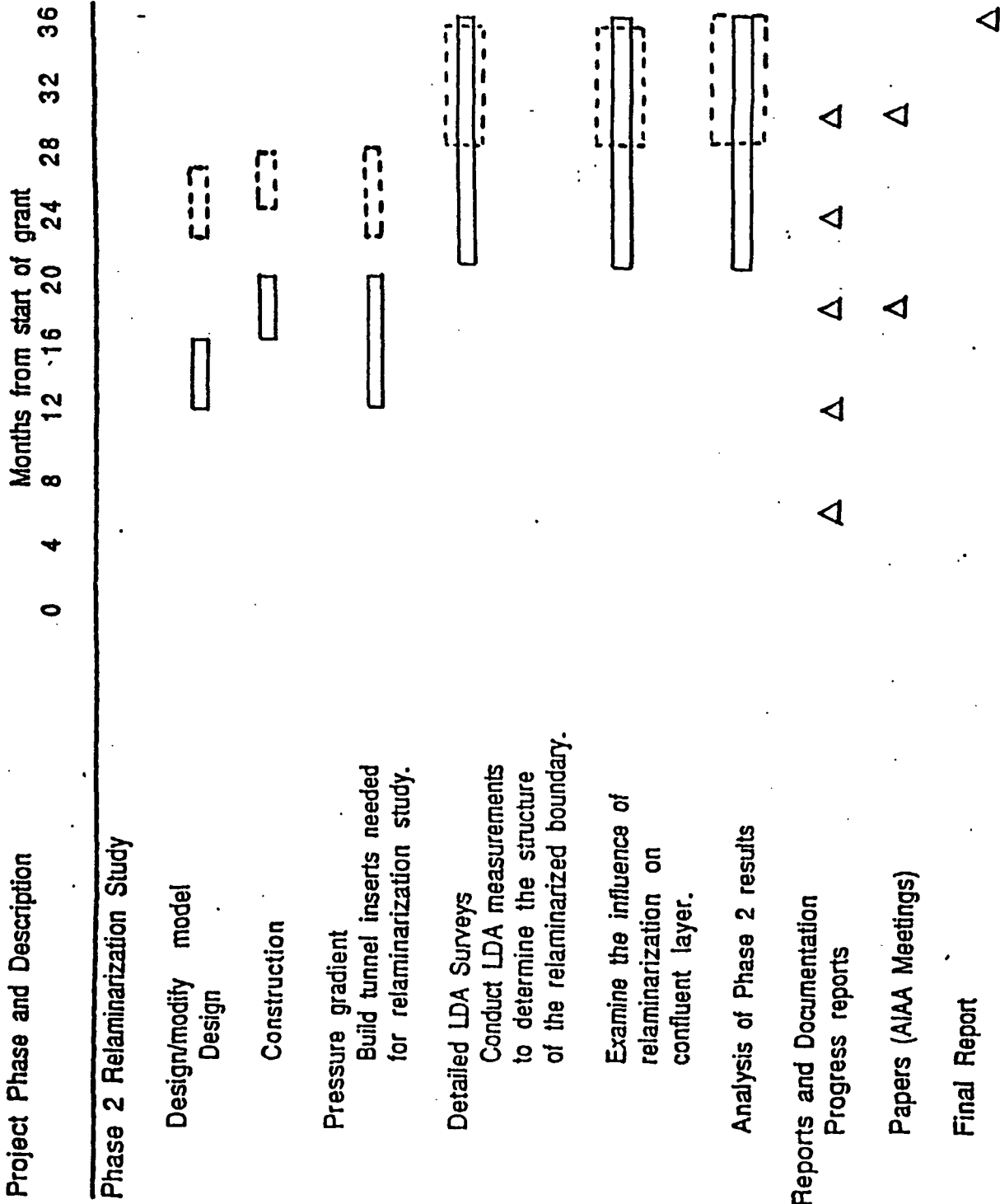
Estimate lift for these cases



Analysis of Phase 1 results



Schedule



The dotted bars indicate revised schedule.

Nondimensional Lift Coefficient Contour Plot  
 (Flap at 13 degree;  $C_{l_{total}}/C_{l_0}$ ;  $C_{l_0}=1.484$ )

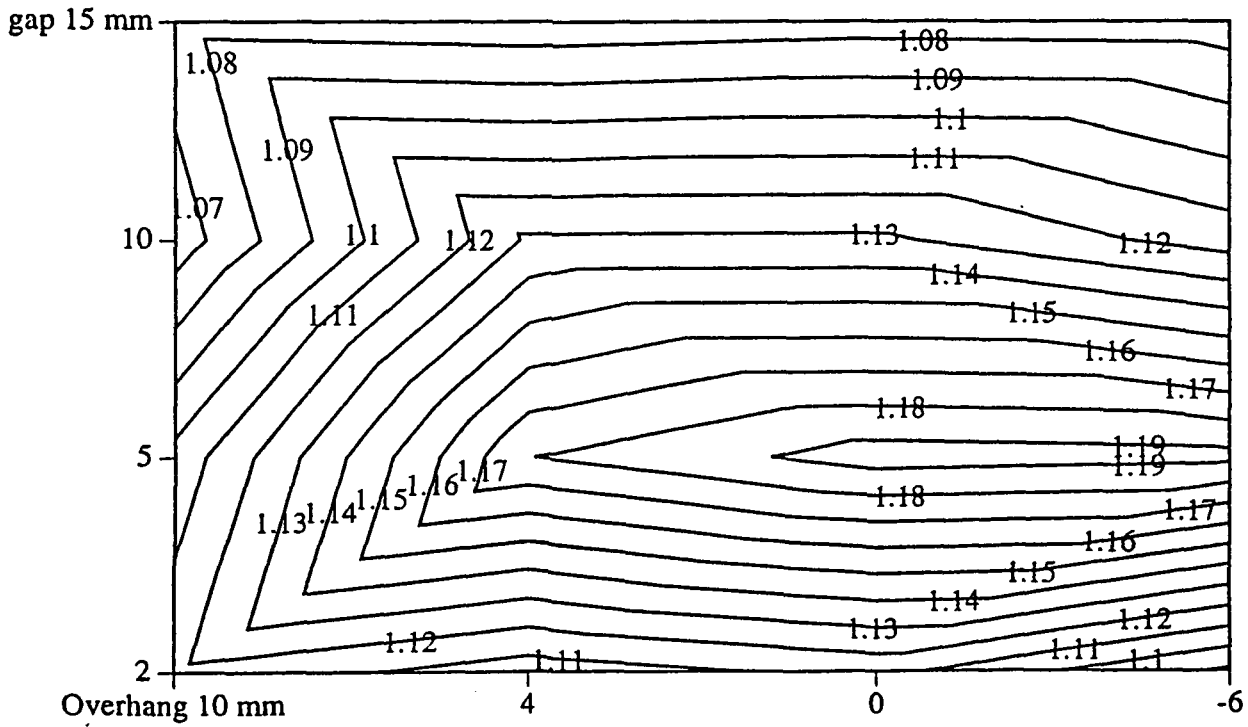
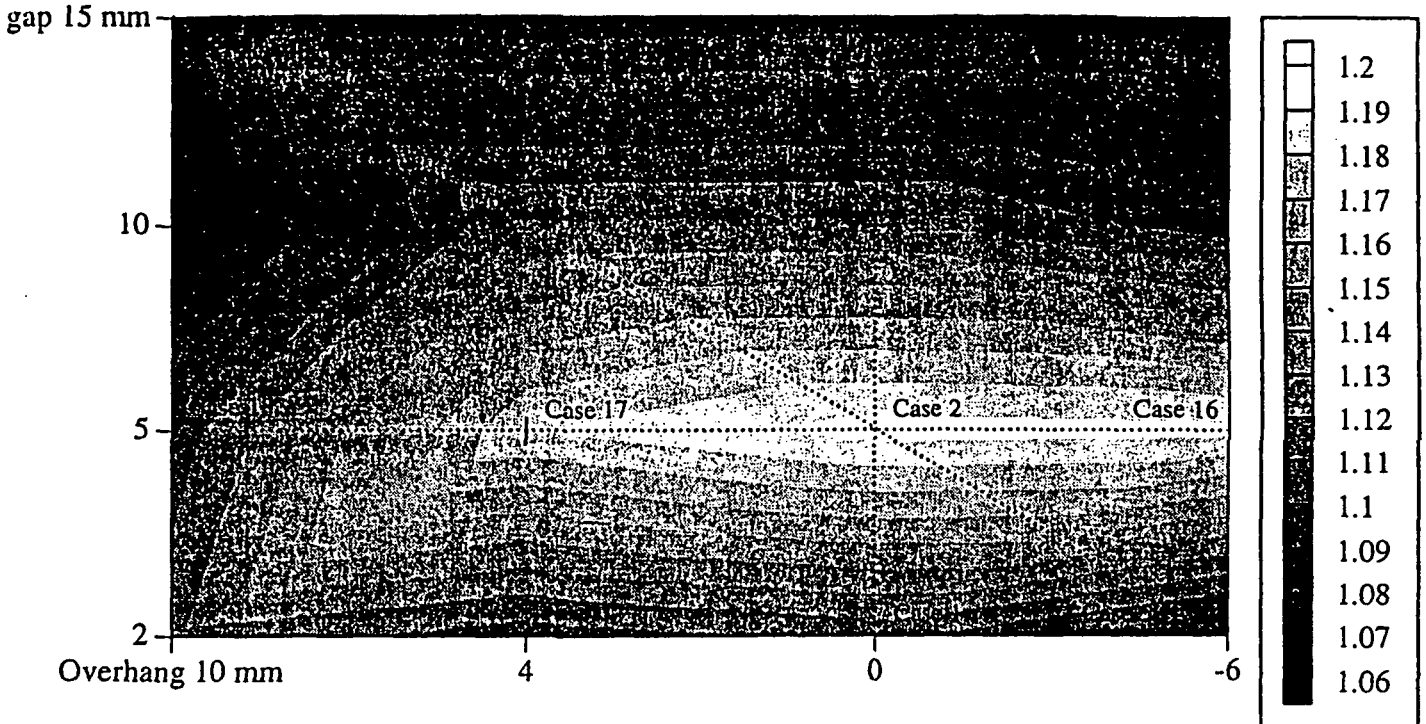
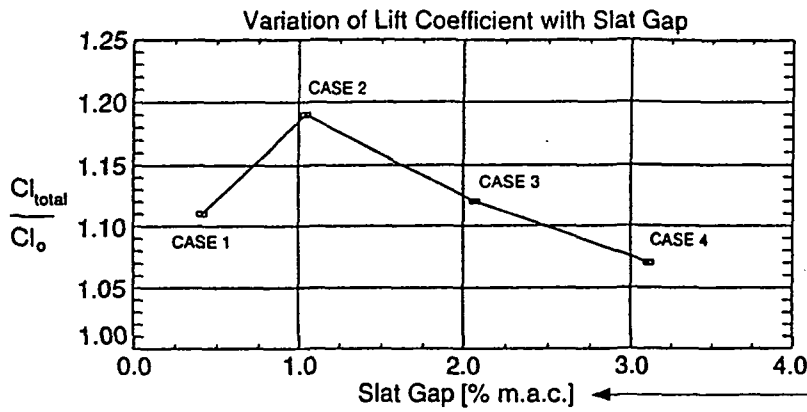
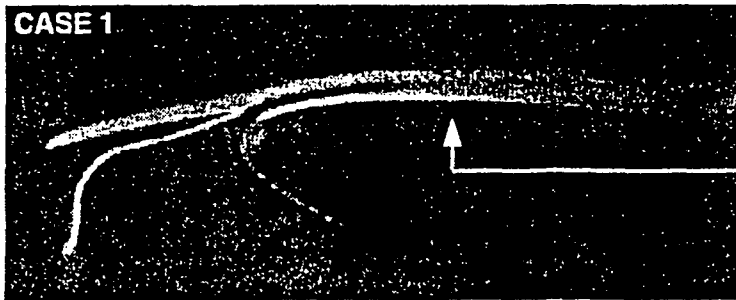


Figure 1

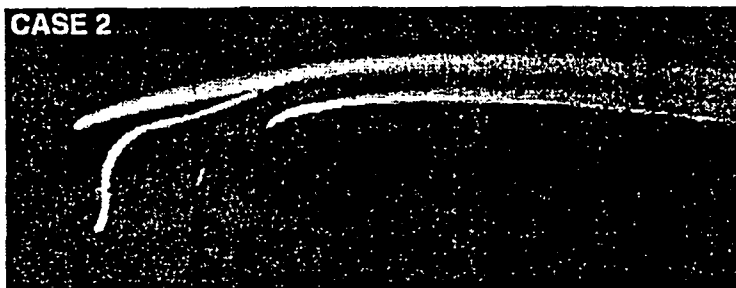


$Re = 500,000$   
 $\delta_f = 13.0^\circ$   
 $\alpha = 10.0^\circ$   
 $\alpha_s = -10.0^\circ$   
 $O_s = 0.0\%$     $O_f = 0.0\%$   
 $G_s = 0.41-3.11\%$     $G_f = 0.0\%$

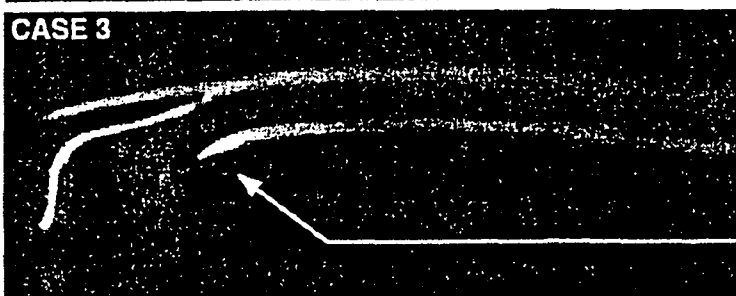


**CASE 1**  
 Slat wake merges with primary airfoil boundary layer

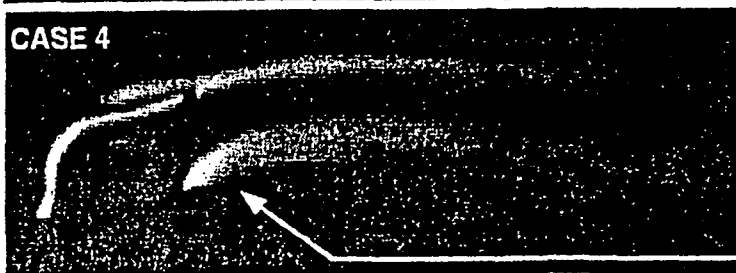
Condition of early confluence



**CASE 2**  
 Point of confluence farther aft: optimum Cl case

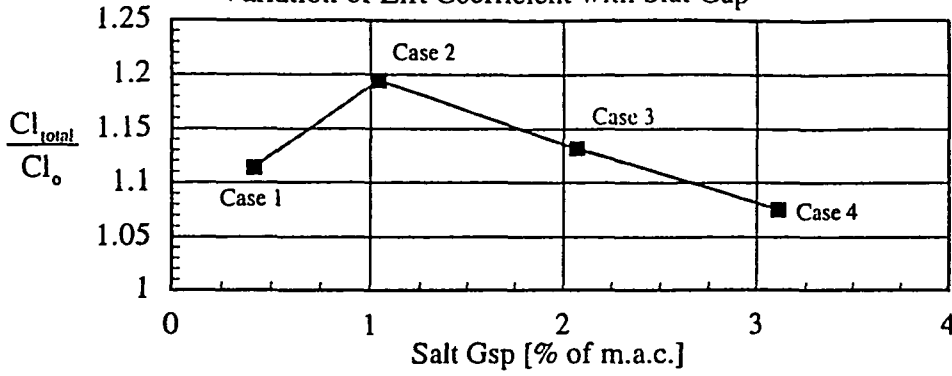


**CASE 3**  
 Separation bubble forms on leading edge

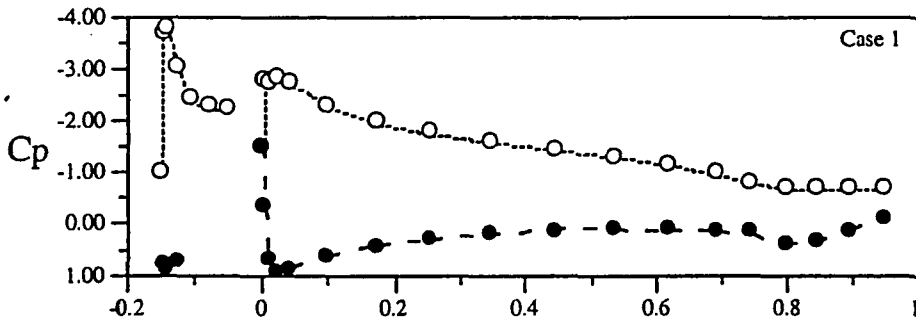


**CASE 4**  
 Separation at leading edge

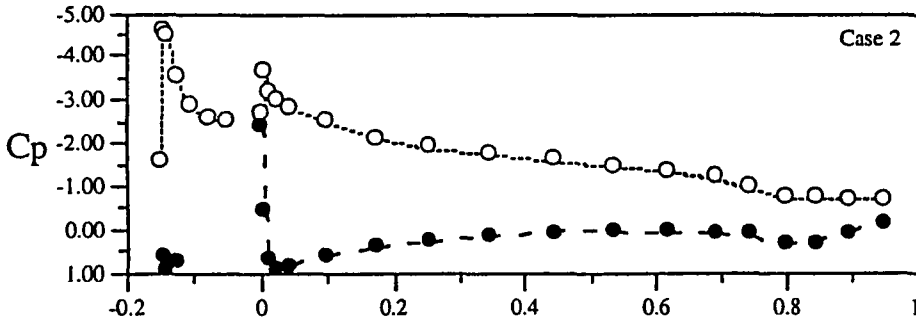
Variation of Lift Coefficient with Slat Gap



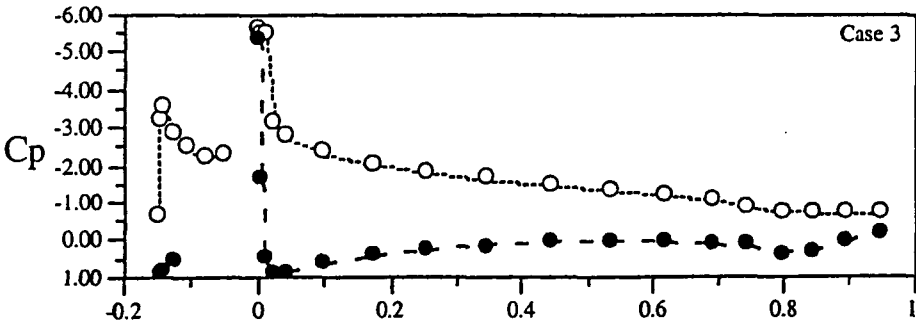
$Re = 1M$   
 $\delta_r = 13^\circ$   
 $\alpha = 10^\circ$   
 $\alpha_s = -10^\circ$   
 $O_s = 0.0\%$



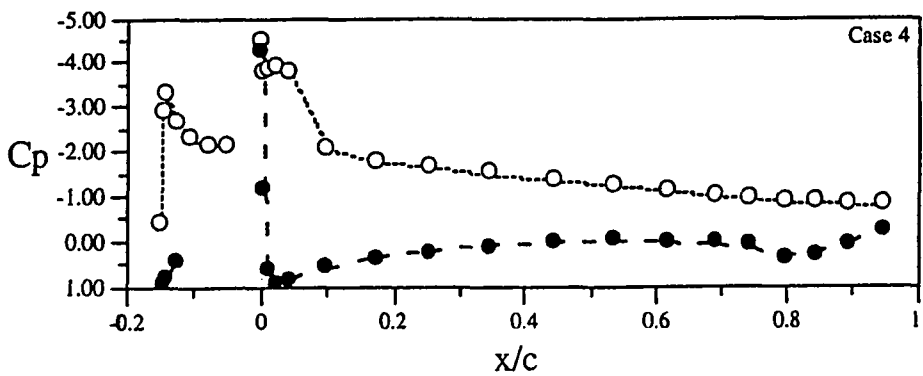
$O_s = 0.0\%$   
 $G_s = 0.41\%$



$O_s = 0.0\%$   
 $G_s = 1.04\%$



$O_s = 0.0\%$   
 $G_s = 2.07\%$



$O_s = 0.0\%$   
 $G_s = 3.11\%$

## Blockage/Wall Interference

Experiments were conducted to assess wind tunnel and wall interference on aerodynamic measurements.

High lift model was tested in both a small and large subsonic wind tunnel.

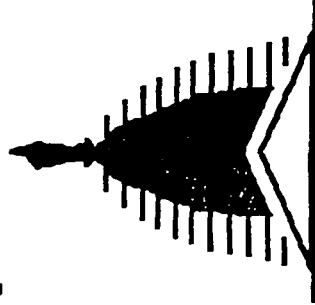
Tunnel cross-sections were 2' x 2' and 5' x 5'.

Model blockage at highest angle of attack

In 2' x 2' tunnel ~ 15%

In 5' x 5' tunnel ~ 2.4%

Standard blockage corrections could be applied to 2' x 2' test section data to correct results to match 5' x 5' test section measurements for model/flap configuration.



UNIVERSITY of  
NOTRE DAME



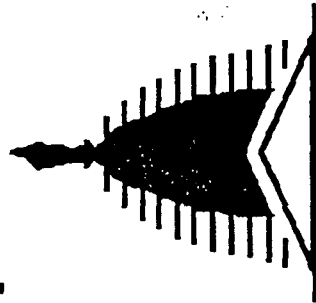
## **Blockage/Wall Interference**

**Pressure distributions were similar. Blockage effect resulted in higher effective angles of attack for model in 2' x 2' tunnel.**

**Integrated lift coefficients showed same trends with gap and overhang in both tunnels.**

**Wing/flap data can be corrected for blockage using standard wind tunnel correction techniques, slat/wing/flap configuration is not easily corrected.**

**Although measurements in the 2' x 2' tunnel are affected by blockage, the aerodynamic and flow characteristics exhibit similar behavior to the experiments in the larger tunnel where blockage is minimal. Our conclusion is that the flow physics is essentially the same in both tunnels and that blockage results in a different effective angle of attack.**



**UNIVERSITY of  
NOTRE DAME**

## **Rationale for Continued Use of Smaller Subsonic Wind Tunnel**

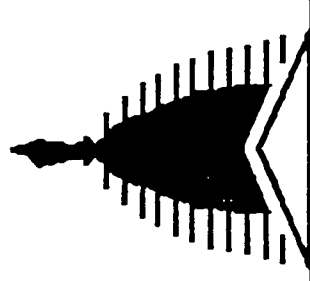
**To study the flow physics we decided to continue our experiments in the 2' x 2' tunnel for the following reasons.**

**Blockage does not affect the flow physics.**

**Aerodynamic characteristics, pressure distributions and influence of gap and overhang similar to data obtained in higher Reynolds number tests.**

**Testing in the smaller tunnel gives us higher resolution for the detailed flow surveys.**

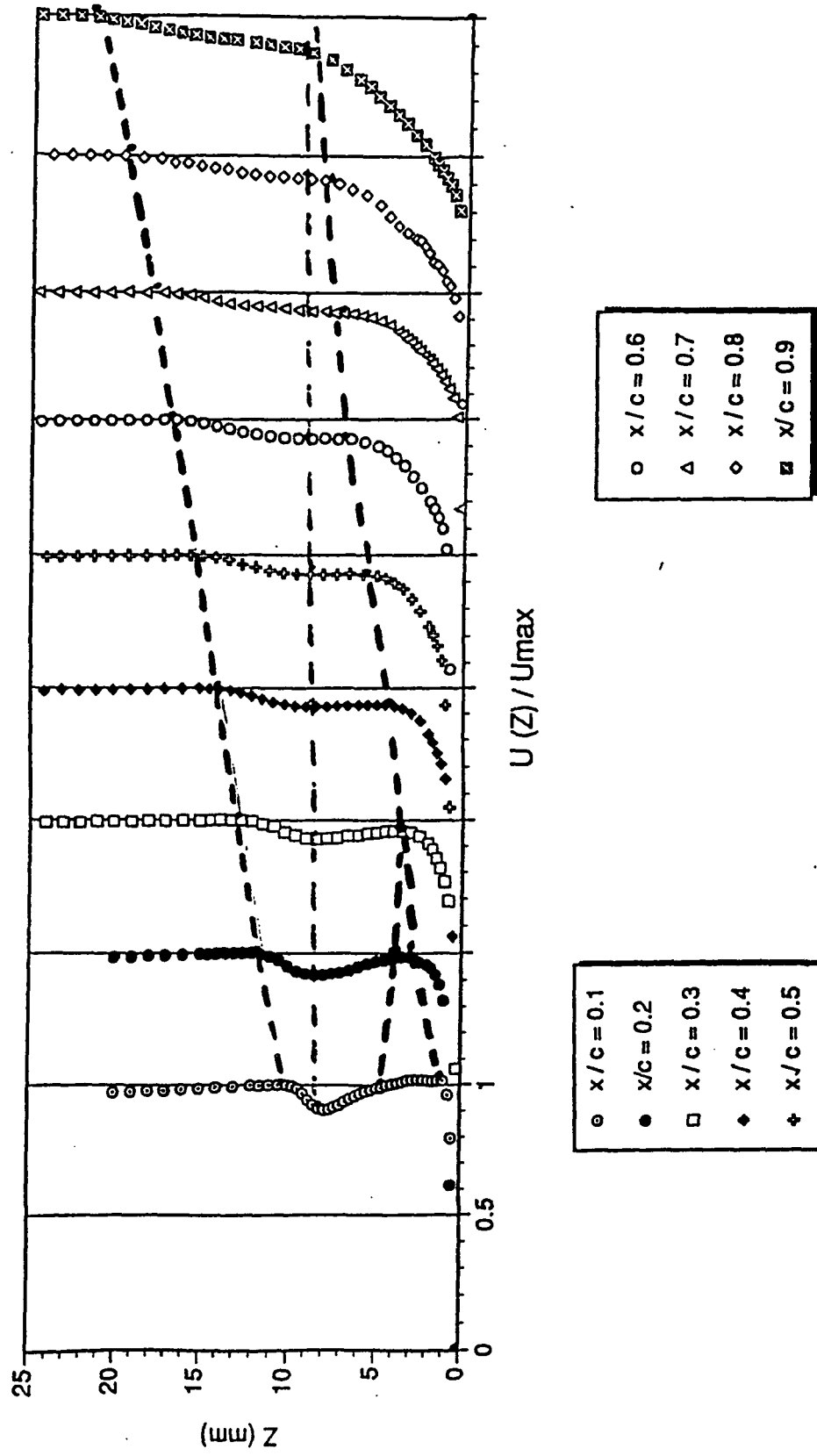
**Flow visualization in the smaller tunnel is superior to the larger tunnel.**



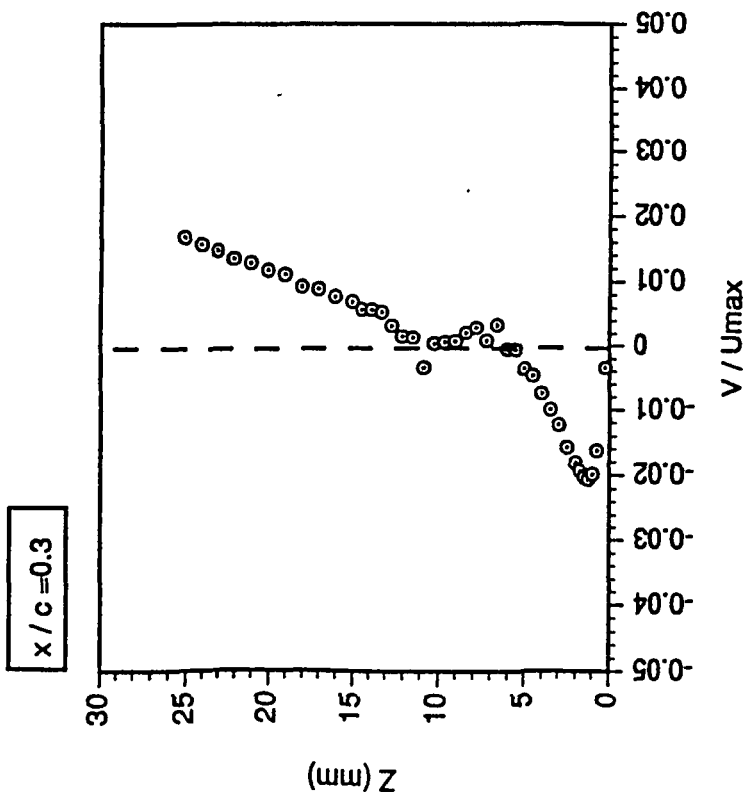
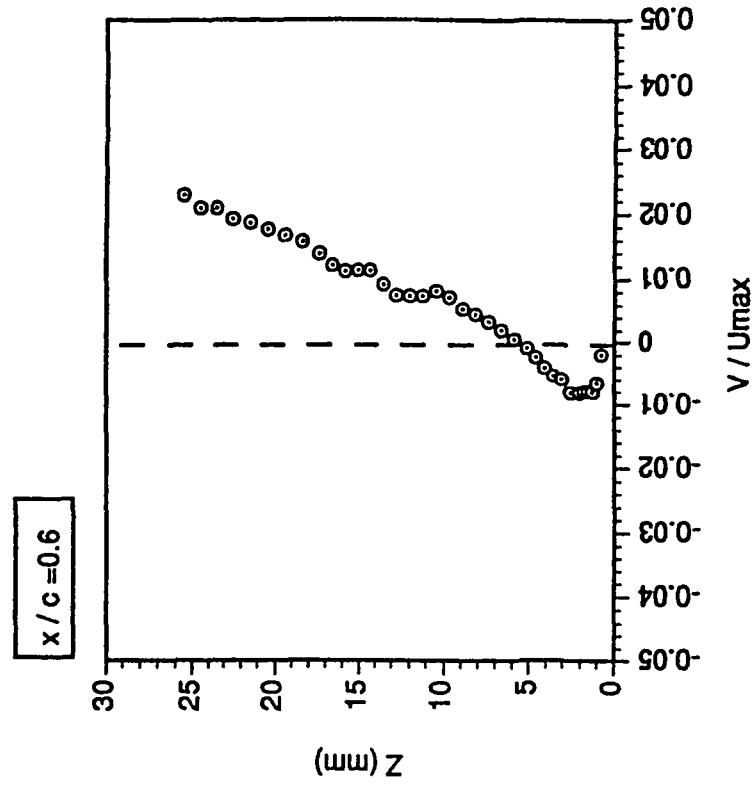
**UNIVERSITY of  
NOTRE DAME**

# LDV Survey : Optimum $CL$ Case

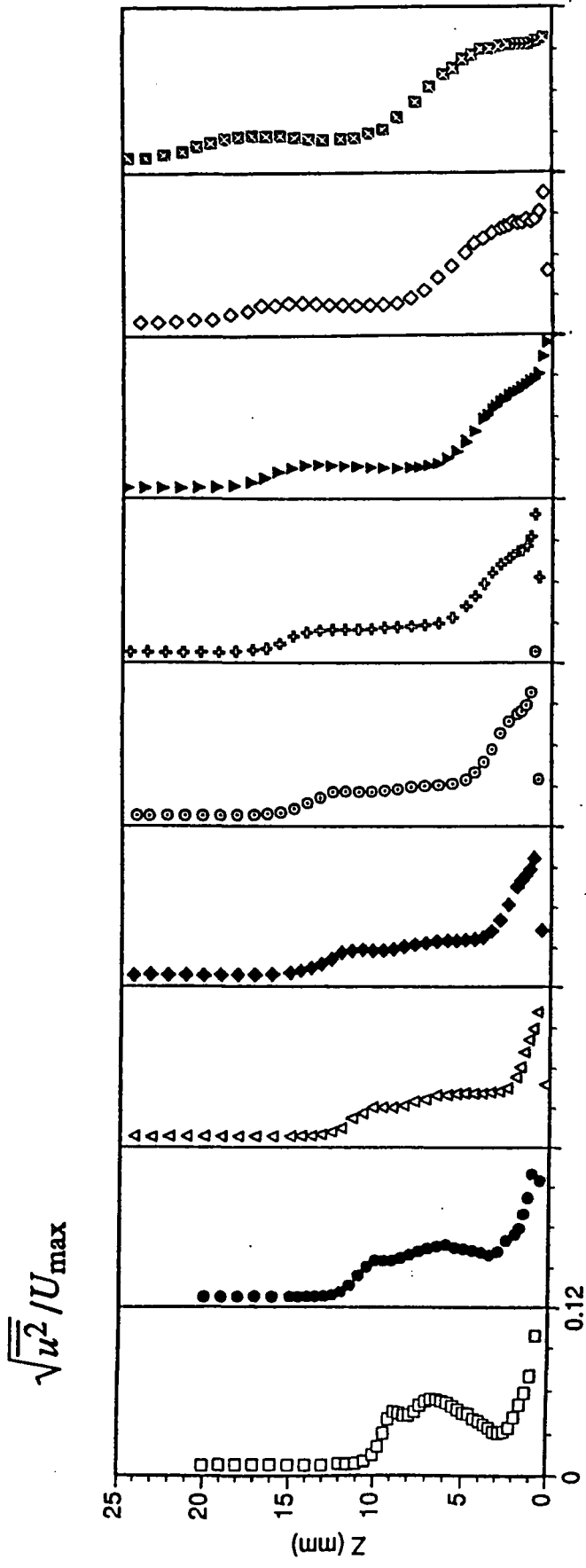
## Mean Velocity Profiles



# LDV Survey Results



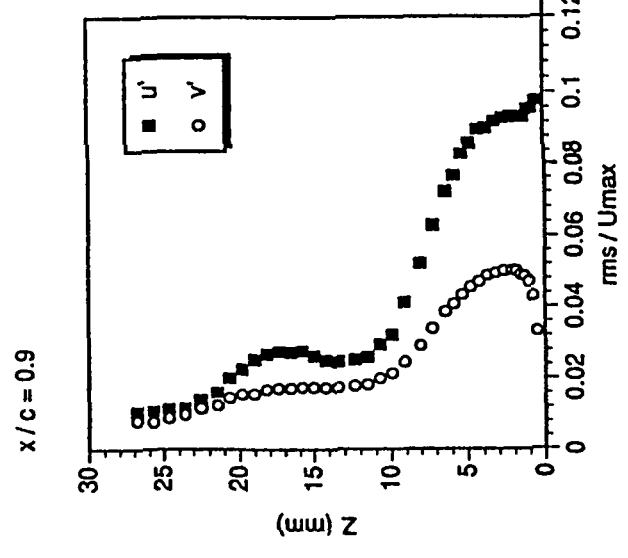
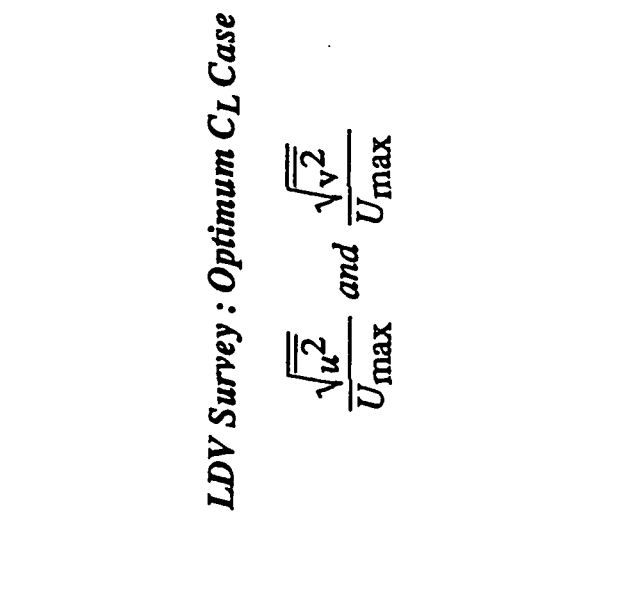
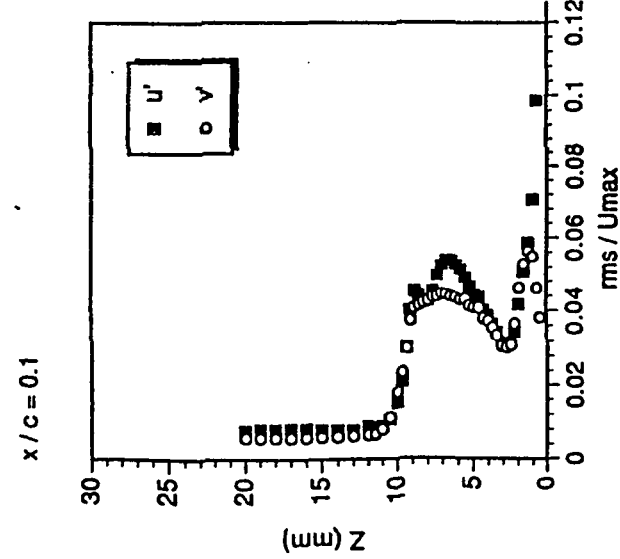
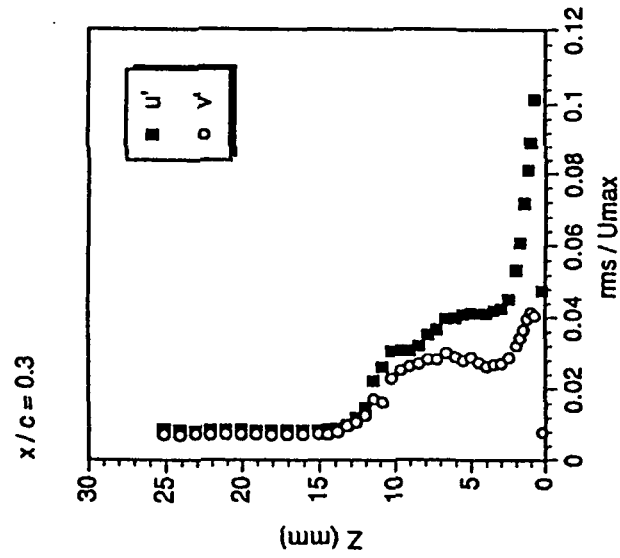
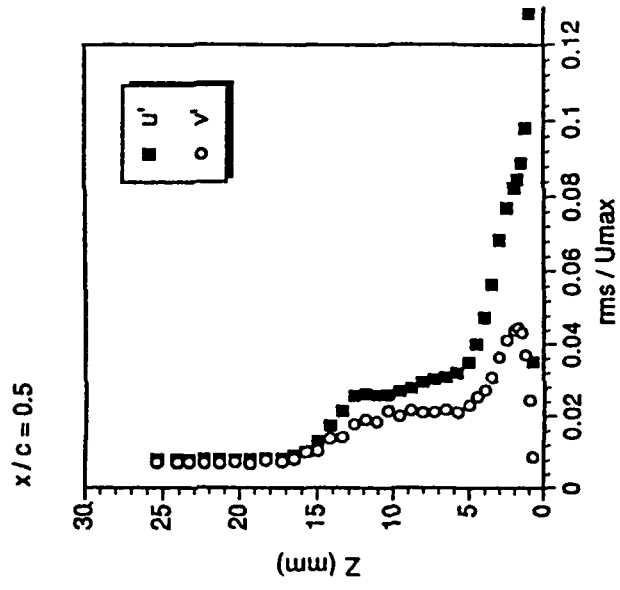
*LDV Survey : Optimum CL Case*



- x/c = 0.1
- x/c = 0.2
- △ x/c = 0.3
- ◆ x/c = 0.4
- ⊙ x/c = 0.5
- ⊕ x/c = 0.6
- ▽ x/c = 0.7
- ◇ x/c = 0.8
- ⊠ x/c = 0.9

u' / Umax

ORIGINAL PAGE IS OF POOR QUALITY

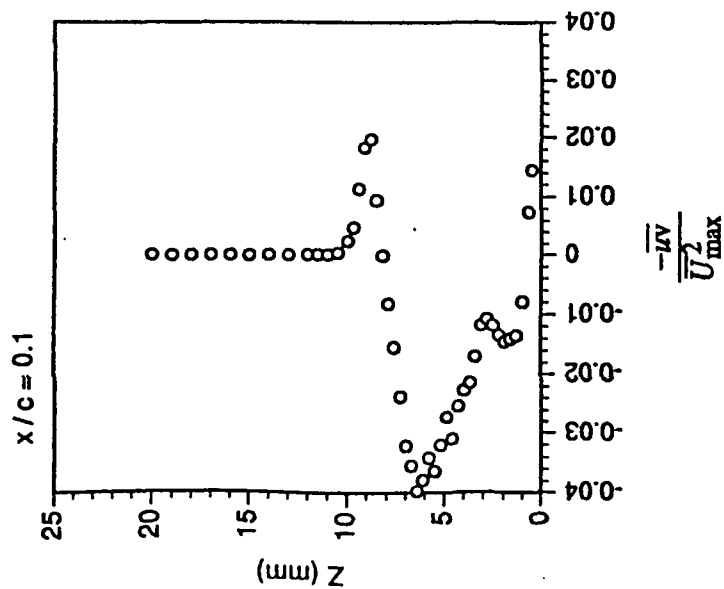
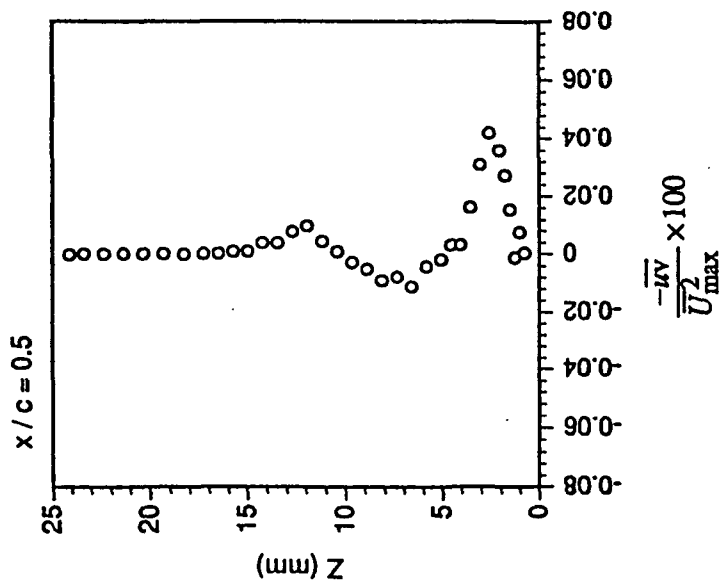
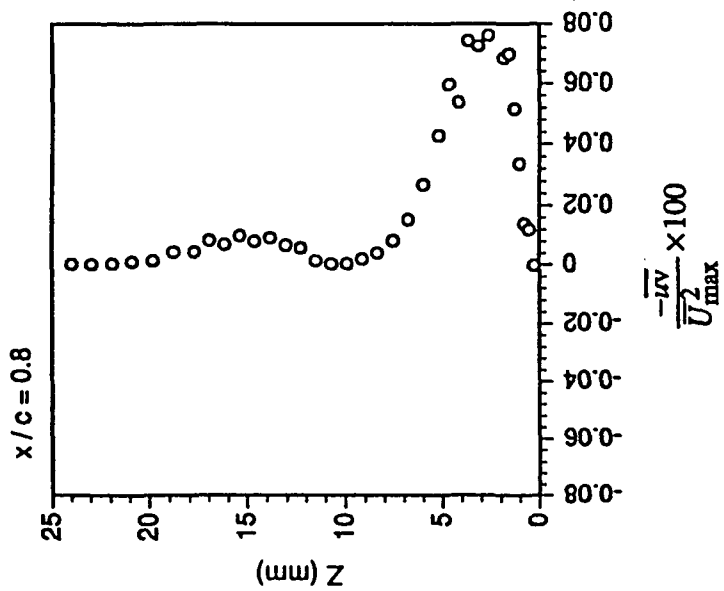


*LDV Survey : Optimum CL Case*

$\frac{\sqrt{u'^2}}{U_{max}}$  and  $\frac{\sqrt{v'^2}}{U_{max}}$

*LDV Survey : Optimum CL Case*

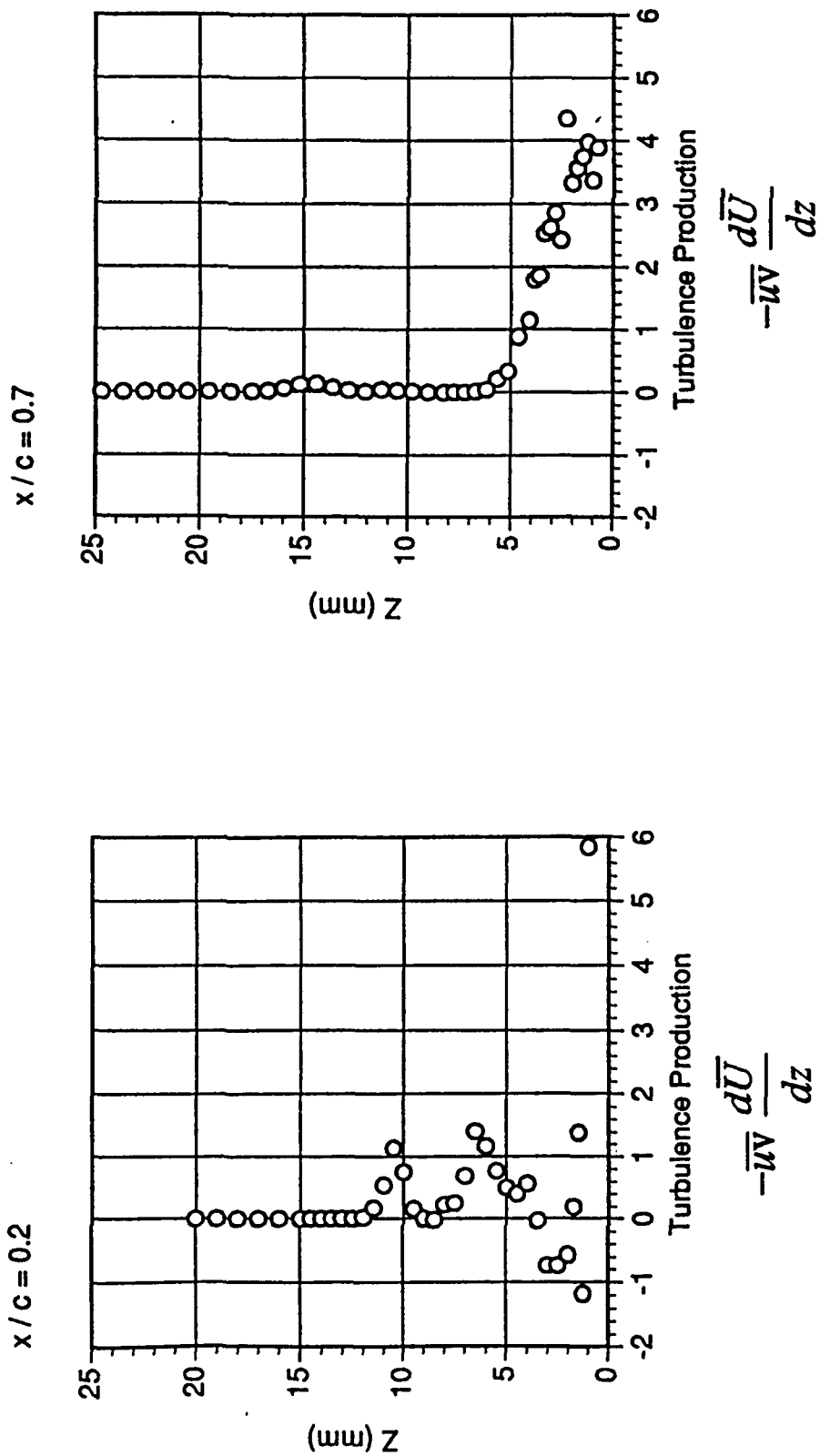
$$\frac{-\overline{UV}}{U_{\max}^2} \times 100$$



ORIGINAL PAGE IS  
OF POOR QUALITY

# LDV Survey : Optimum CL Case

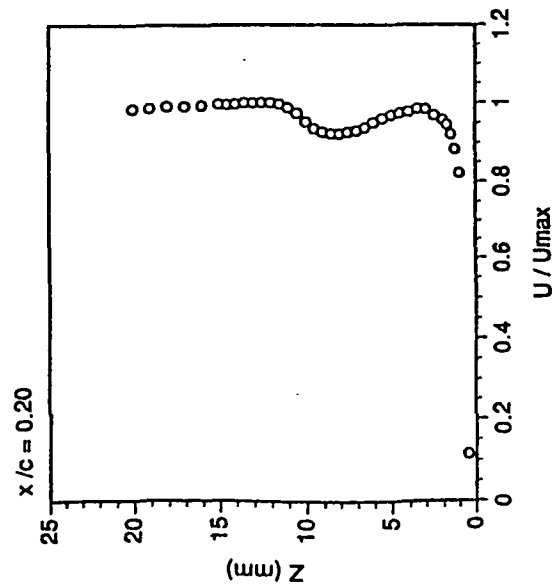
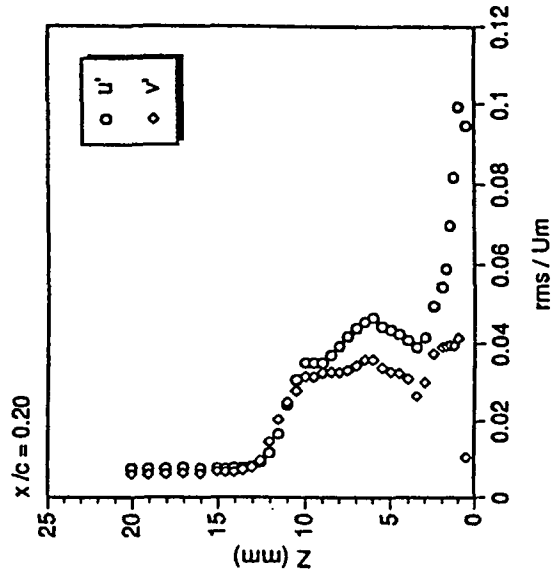
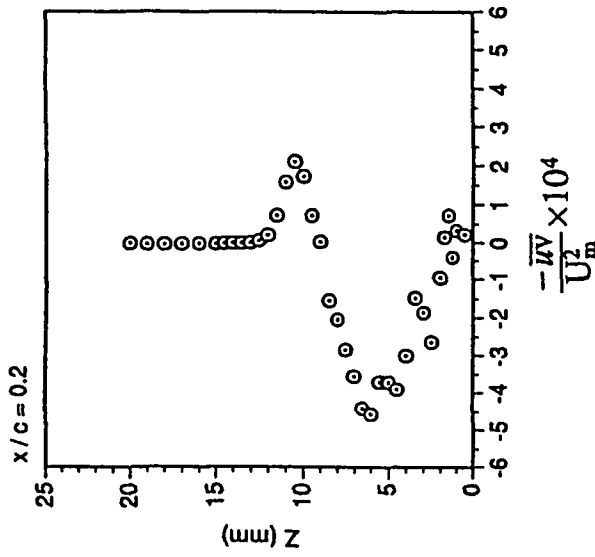
## Turbulence Production





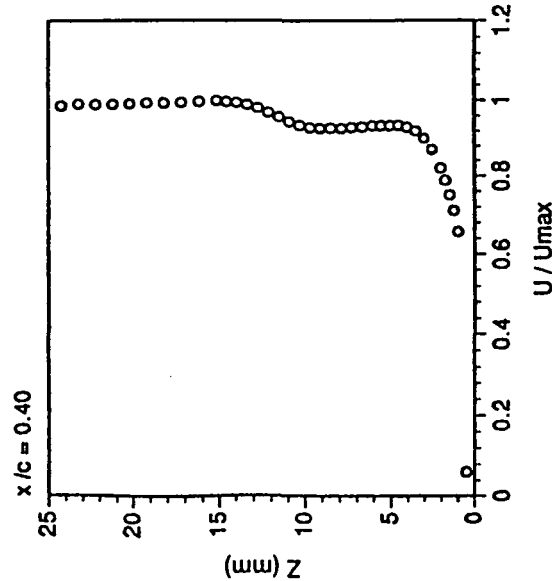
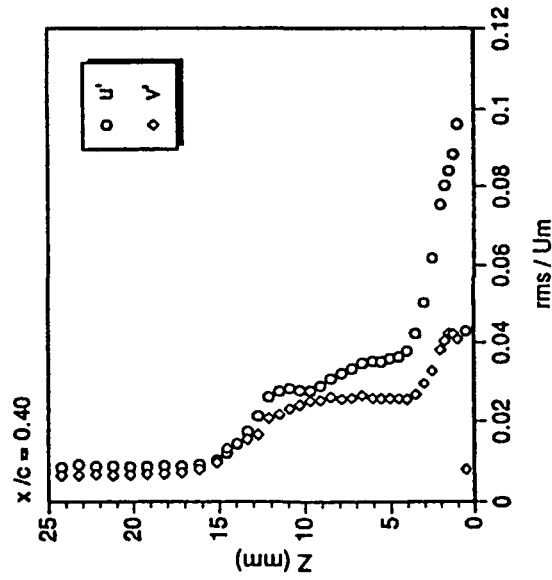
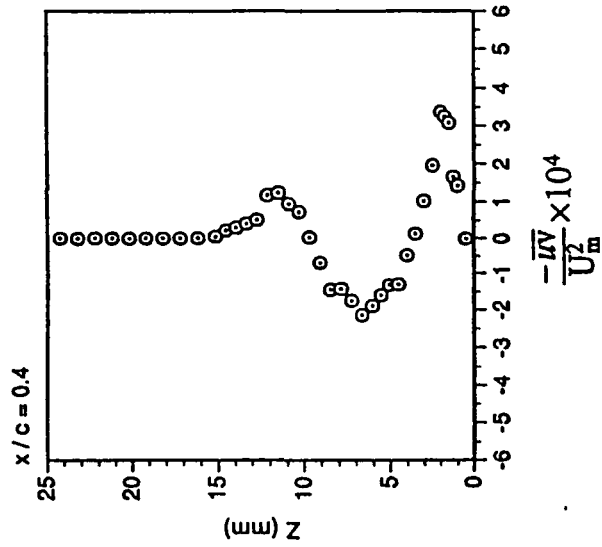
# LDV Flow Survey :

## Optimum $C_{Lmax}$ Case



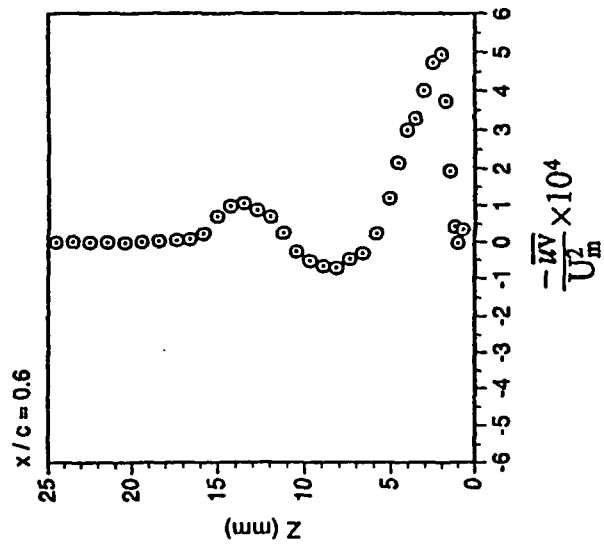
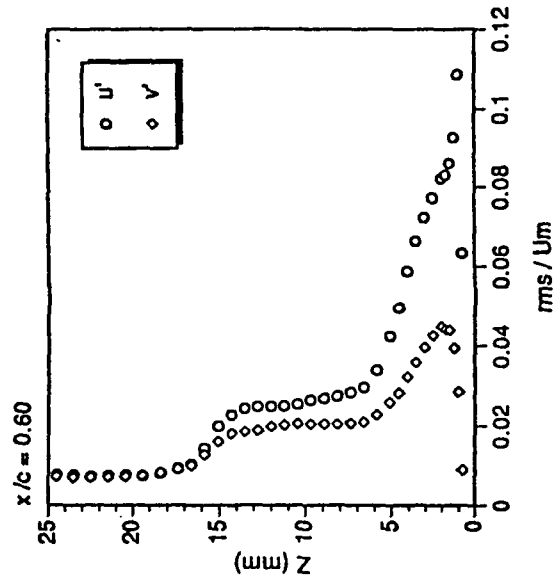
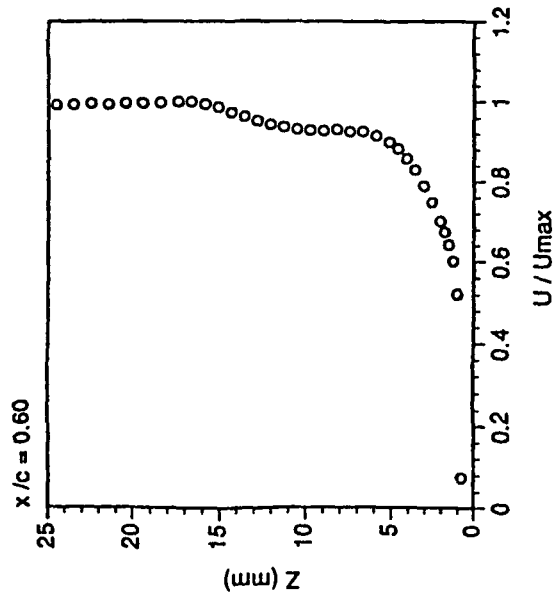
# LDV Flow Survey:

## Optimum $C_{Lmax}$ Case



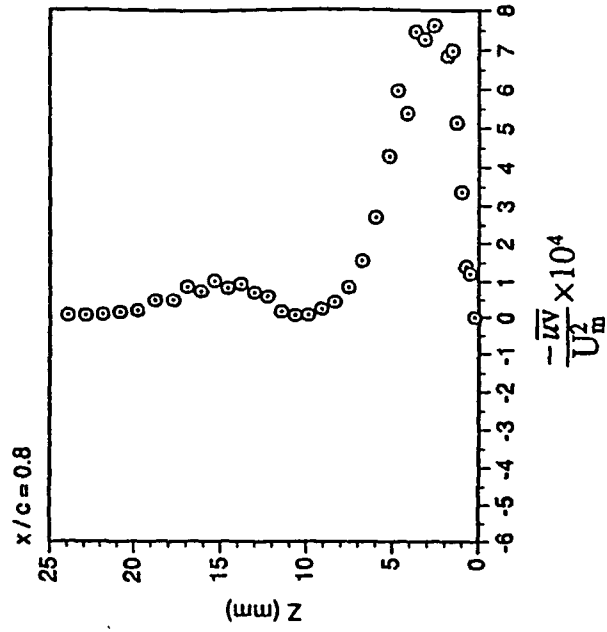
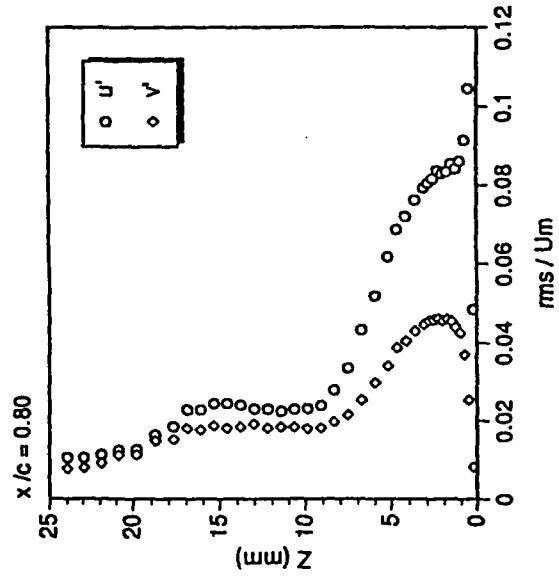
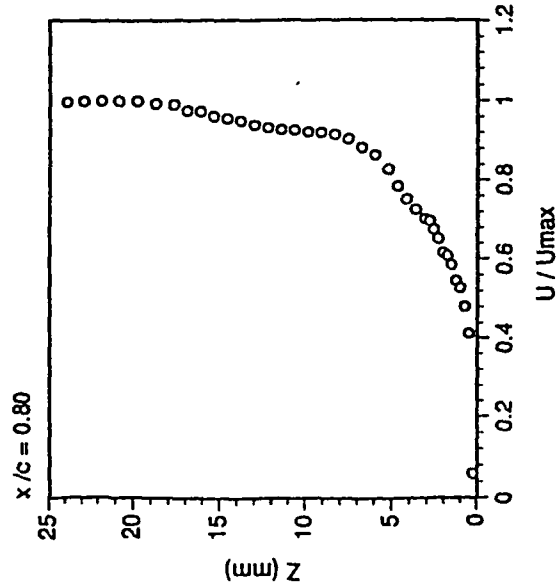
# LDV Flow Survey :

## Optimum $C_{Lmax}$ Case



# LDV Flow Survey :

## Optimum $C_{Lmax}$ Case



**Additional Information Gleaned from  
Detailed LDV Surveys**

- **Turbulent Structure of Confluent Boundary Layer**
- **Slat Wake Structure (widening, asymmetry, etc. )**
- **Turbulence Production**
- **Turbulent Kinetic Energy**
- **Mixing length distribution (memory effects)**
- **Energy Balance: Examination of terms relevant to k- $\epsilon$  modelling; Significance of pressure diffusion**
- **Effect of Streamline Curvature**
- **Effect of Relaminarization**

# *Research Goals*

---

- Complete LDV Study of Optimum  $C_L$  Case.
- Perform Detailed LDV Confluent Boundary Layer Surveys for Multiple Non-Optimum  $C_L$  Cases. (Essential for Modeling).
- Obtain Skin Friction Distributions for Both Optimum and Non-Optimum  $C_L$  Cases for Scaling Purposes.
- Data Analysis and Inner and Outer Variable Scaling.
- Set-Up and Perform Relaminarization Experiments
- Final Report Establishing the Role of Leading Edge Confluent Boundary Layer Flow Physics on High Lift Performance.

# Transformation of Confluent Boundary Layer (From High-Lift Model to Flatplate)

