N94-33528

CODE VALIDATION FOR THE SIMULATION OF SUPERSONIC VISCOUS FLOW ABOUT THE F-16XL

51-02
11986

Jolen Flores, Eugene Tu and Lyndell King
Applied Computational Fluids Branch
Fluid Mechanics Laboratory
NASA Ames Research Center
Moffett Field, California 94035

Presentation at the 1st HSRP Workshop
May 14-16, 1991
Williamsburg, VA

INTRODUCTION

Because of the large potential gains related to laminar flow on the swept wings of supersonic aircraft, recent interest in the application of laminar flow control (LFC) techniques in the supersonic regime has increased. A supersonic laminar flow control (SLFC) technology program is currently underway within NASA. The objective of this program is to develop the data base and design methods that are critical to the development of laminar flow control technology for application to supersonic transport aircraft design. Towards this end, the program integrates computational investigations currently underway at NASA Ames-Moffett and NASA Langley with flight-test investigations being conducted on the F-16XL at the NASA Ames-Dryden Research Facility in cooperation with Rockwell International.

The computational goal at NASA Ames-Moffett is to integrate a thin-layer Reynolds averaged Navier-Stokes flow solver with a stability analysis code.[1] The flow solver would provide boundary-layer profiles to the stability analysis code which in turn would predict transition on the F-16XL wing. To utilize the stability analysis codes, reliable boundary-layer data is necessary at off-design cases. Previously, much of the prediction of boundary-layer transition has been been accomplished through the coupling of boundary-layer codes with stability theory.[2,3] However, boundary-layer codes may have difficulties at high Reynolds numbers, of the order of 100 million, and with the current complex geometry in question. Therefore, a reliable code which solves the thin-layer Reynolds averaged Navier-Stokes equations is needed.

The objective of the current research is two-fold. The first objective is method verification, via comparisons of computations with experiment, of the reliability and robustness of the code. To successfully implement LFC techniques to the F-16XL wing, the flow about the leading edge must be maintained as laminar flow. Therefore, the second objective is to focus on a series of numerical simulations with different values of α and Reynolds numbers. The purpose of the simulations is to study their effects on the two main factors which precipitate transition to turbulence at leading edges of highly swept wings (e.g. "spanwise contamination" and "crossflow instability"). The bulk of this presentation will focus on the first stated objective.

CNS BACKGROUND

The Compressible Navier-Stokes (CNS) code is utilized in this research. The CNS code is a time-dependent Navier-Stokes solver implemented in a zonal methodology. The zonal approach allows grids for complex configurations to be generated in topologically simple pieces and patched together to form the global mesh. In addition to simplifying the grid generation process, the zonal approach enhances computational efficiency by allowing zones to involve different physical models so that only the complexity necessary for the local flow field is assumed. The zonal method also gives the user flexibility in allowing different convergence strategies to be used in different zones.

Characteristics of the integration scheme ARC3D are given. The algorithm uses central-differencing in all three directions. Second and fourth order artificial dissipation is added both explicitly and implicitly for stability considerations. The inversion process involves inverting only scalar penta-diagonals. The Baldwin-Lomax model is used to model turbulence viscosity in the thin-layer Reynolds-averaged Navier-Stokes equations.[4]

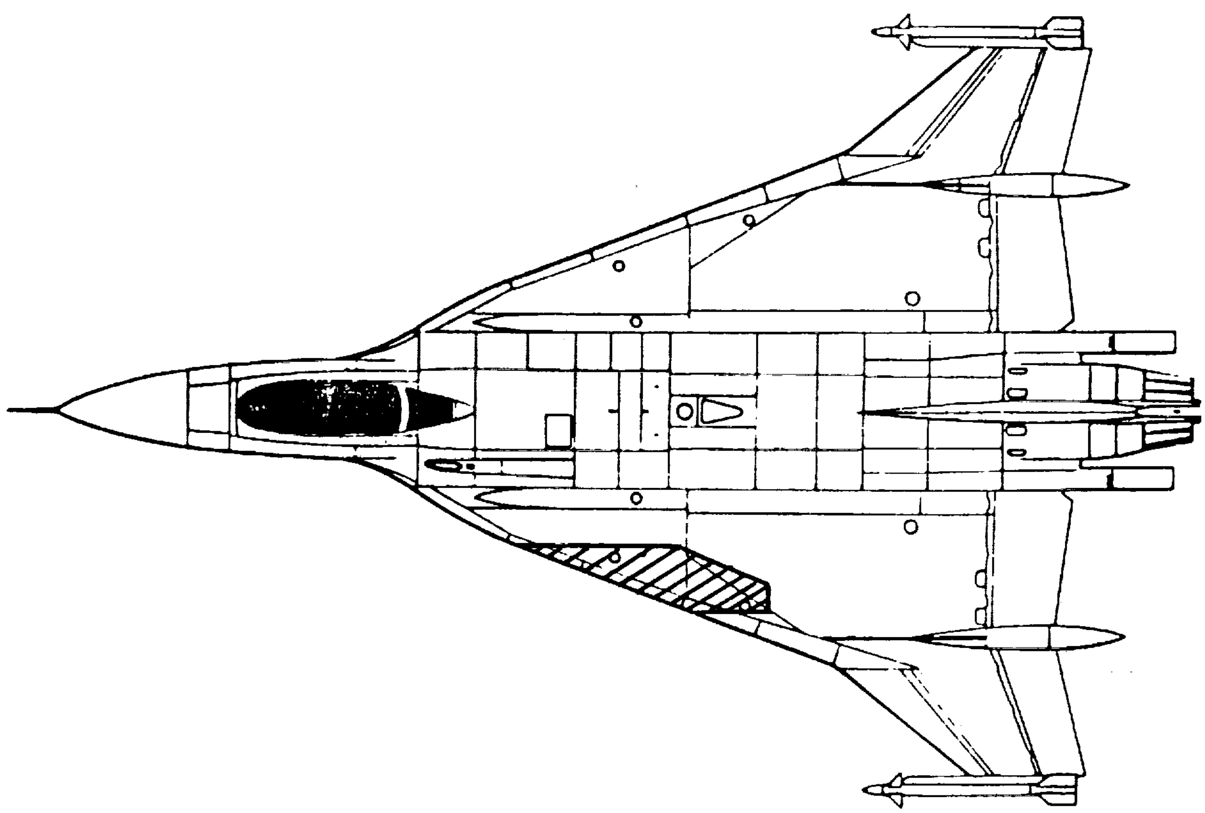
CNS CODE CHARACTERISTICS

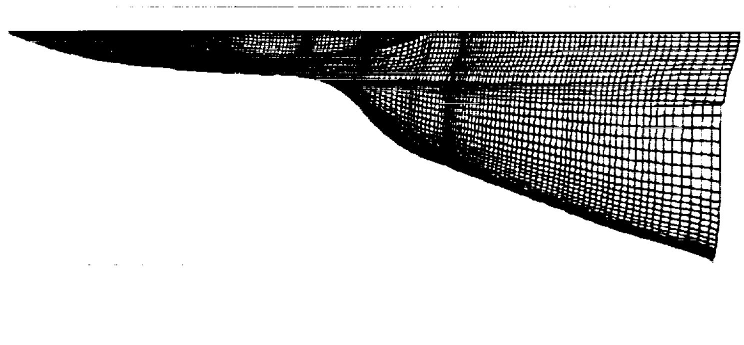
ZONAL SCHEME
- SIMPLIFY GRID GENERATION FOR COMPLEX GEOMETRIES
- COMPUTATIONAL FLEXIBILITY AND EFFICIENCY

ARC3D ALGORITHM
- CENTRAL-DIFFERENCED SCHEME IN ALL THREE DIRECTIONS
- 2ND AND 4TH ORDER EXPLICIT AND IMPLICIT DISSIPATION
- BALDWIN-LOMAX TURBULENCE MODEL

GEOMETRY AND GRID

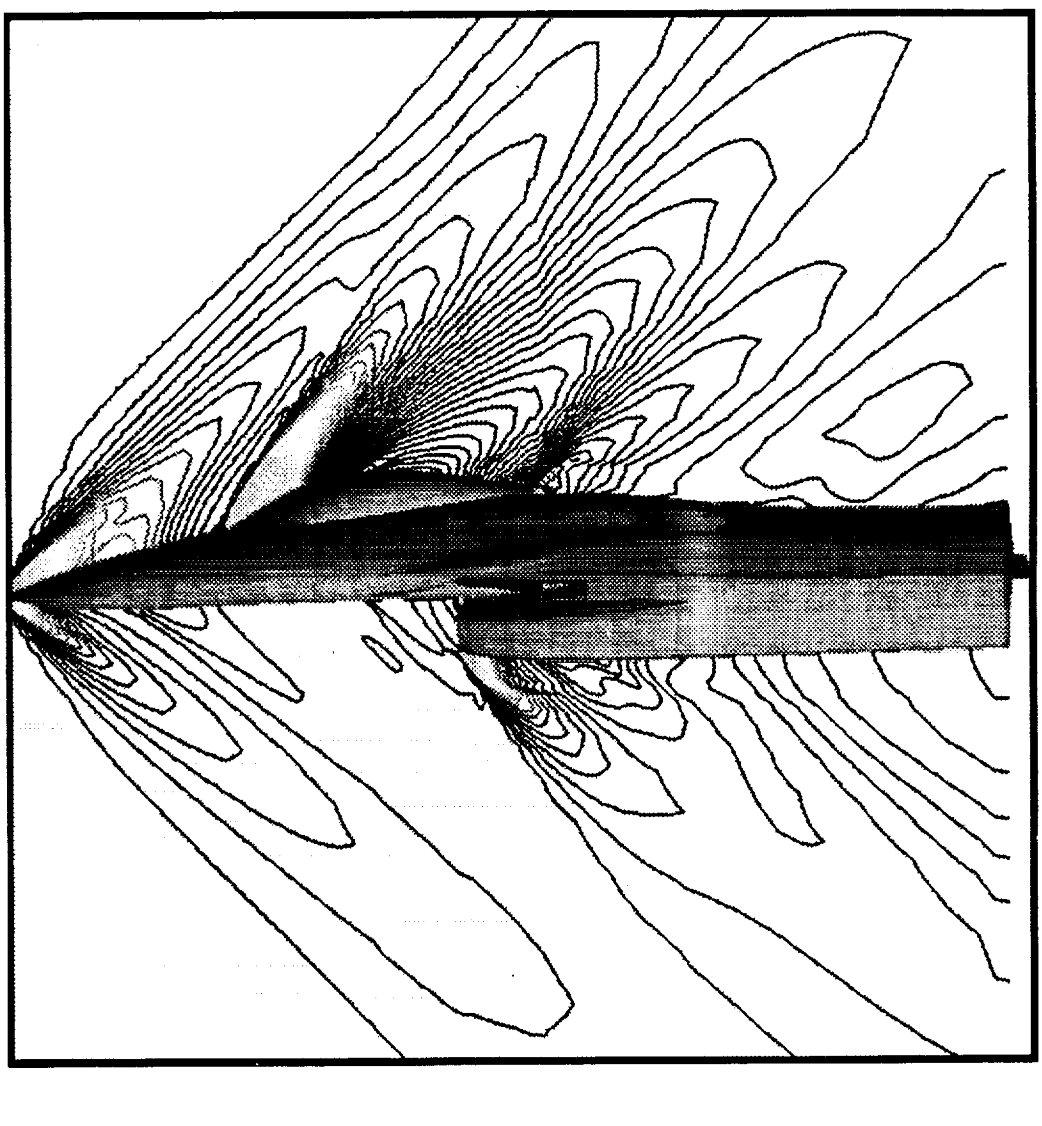
The geometry used for the SLFC program is the F-16XL configuration. It is basically an F-16A with the original wing replaced with a double delta-wing having a sweep angle of 70°, forward of the wing-break. The figure shows a planform view of the surface grid used in the computations. The surface and flow field grids were graciously provided by Dr. C. J. Woan of Rockwell International. Not shown, but modeled, are the inlet, diverter, and environmental control system on the underside of the geometry. Instrumented on the actual flight configuration is a fitted glove on the upper surface of the wing. The glove surface contains tiny holes, created by laser beams to provide suction as a means of maintaining laminar flow. The approximate location of the glove geometry is shown in the figure.





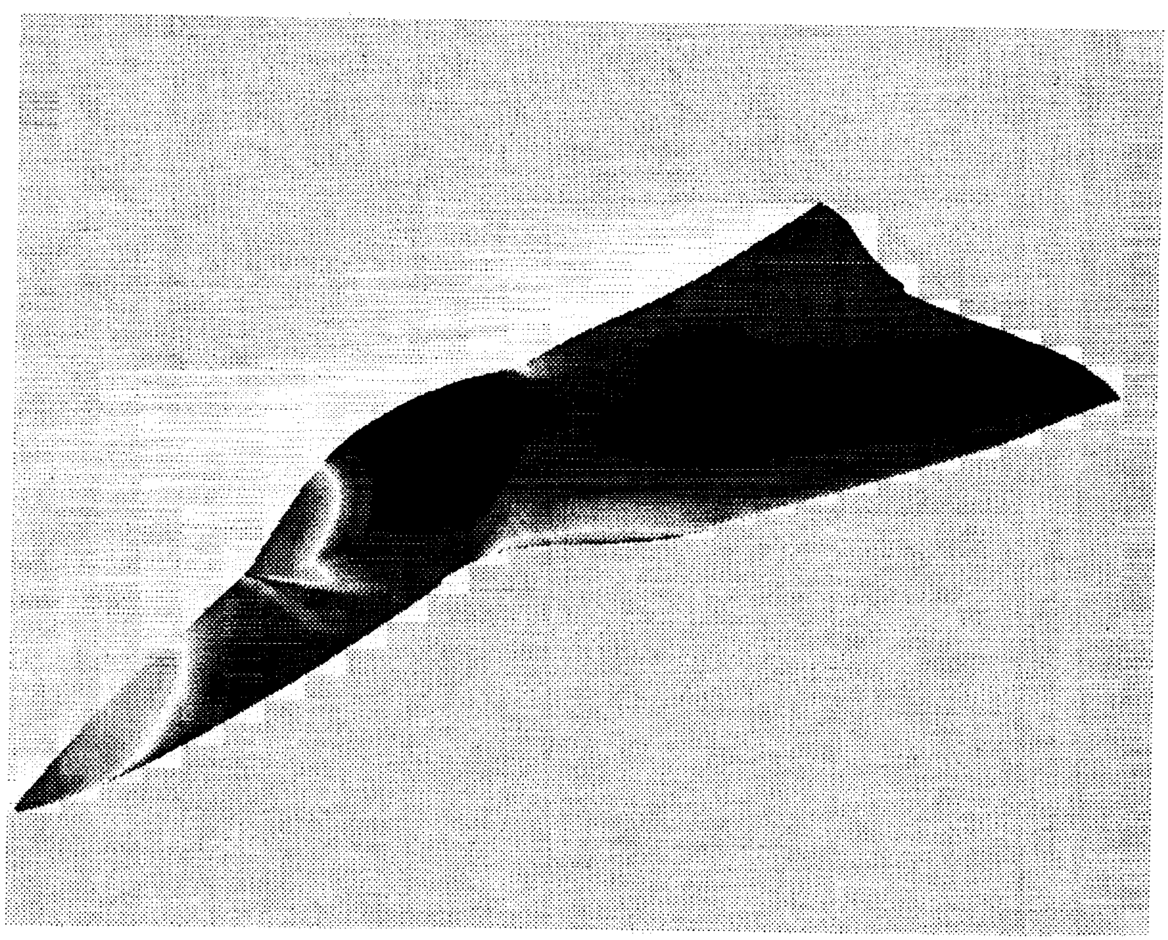
SYMMETRY PLANE PRESSURE CONTOURS

The numerical simulation was conducted with flow conditions approximately matching flight conditions at $M_\infty = 1.6, \alpha = 2.0^\circ$ and $Re_L = 116$ million. The Reynolds number is based on the fuselage length, which is approximately 550 inches. Nineteen zones were used for the computation with a total of one million grid points. The computation required approximately 2500 iterations to drop the initial L2-norm in each zone by three orders of magnitude. On the NASA supercomputer, this required approximately 13 hours of cpu time. The figure illustrates the pressure contours on the symmetry planes. Shocks can be seen at the nose, canopy and lip of the inlet on the geometry. What can also be noted is the smoothness of the contours, even though they are traversing different zones. An expansion wave at the top of the canopy, as well as a recompression shock at the back of the canopy, can also be seen. These regions cause adverse pressure gradients which can cause the flow to separate.



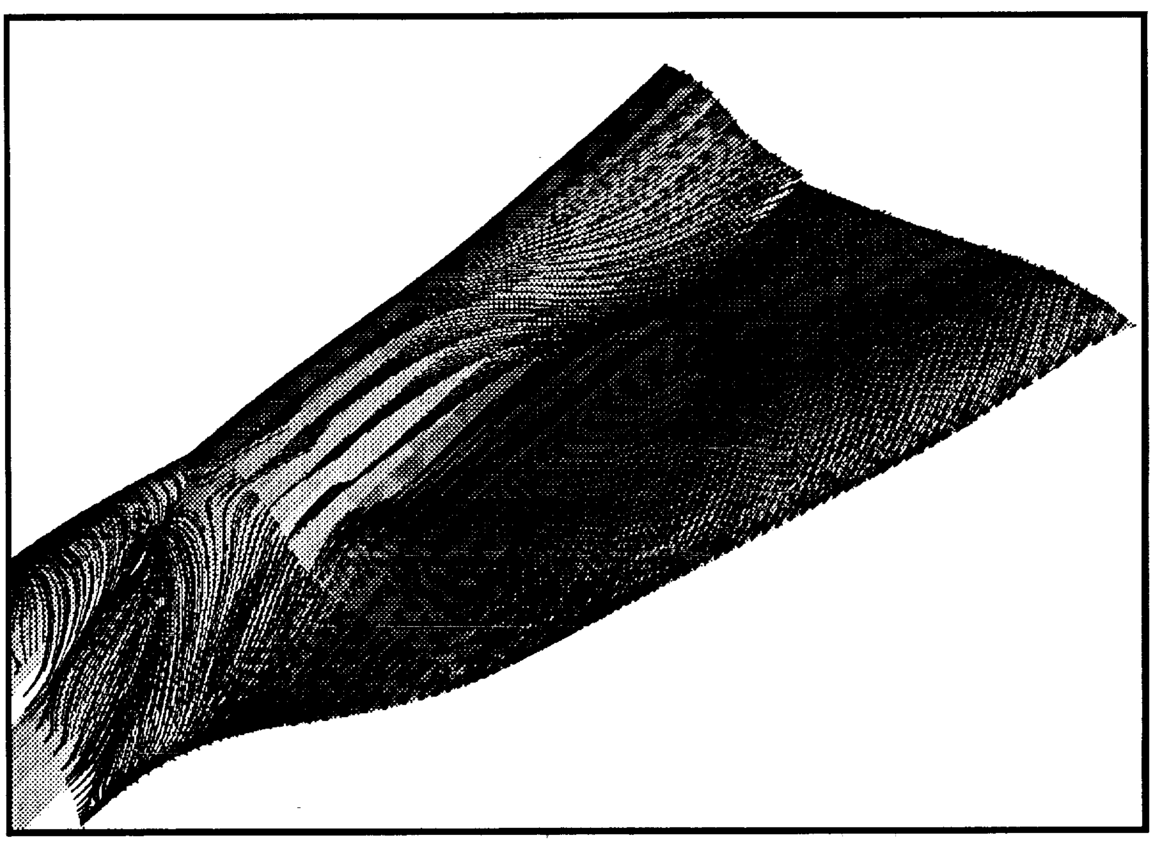
SURFACE PRESSURE MAP

The surface pressure map is illustrated in this figure. Again, the "hot spots" (light shaded regions) at the nose and front of the canopy are noted. The low pressure region (dark shaded regions) at the top of the canopy is also seen and is due to the expansion of the flow about the canopy. A large low pressure region is also seen on the wing of the geometry. It will be shown that this region will have a large influence on the flow pattern in this area.



SIMULATED OIL FLOW PATTERN

Oil flow patterns on the geometry are simulated by releasing and restricting particles to one grid point off the surface and tracking their subsequent journey downstream. As can be seen, a separation region occurs due to the recompression shock at the back of the canopy, however it quickly reattaches downstream. The low pressure region at the top of the canopy causes an upwash of the flow about the fuselage-strake region. Also the flow just off the symmetry plane near the back of the fuselage is seen to be pulled down onto the wing due to the aforementioned low pressure on the wing. This same low pressure also causes the flow coming from the leading edge to head slightly inboard before proceeding downstream.

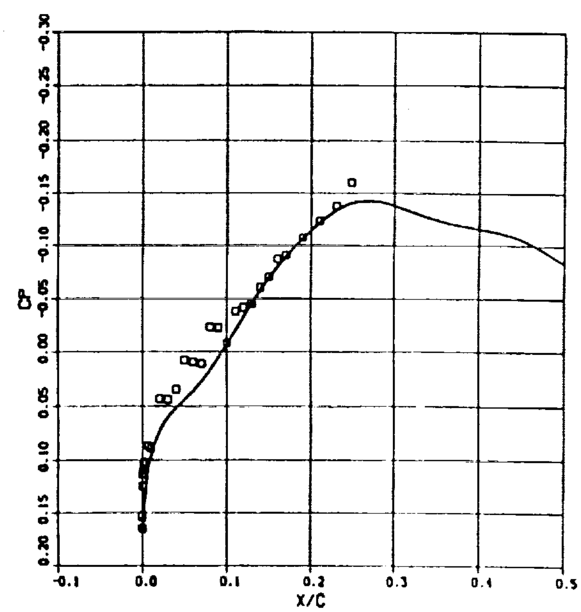


PRESSURE COEFFICIENT COMPARISIONS

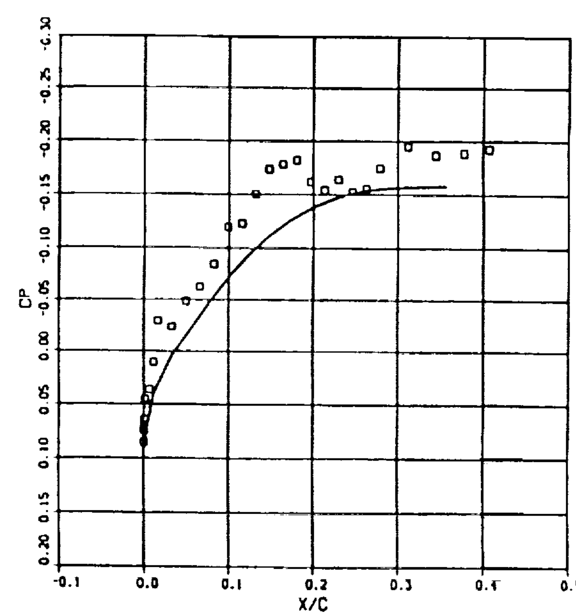
Computed pressure coefficients are compared to inflight data obtained from NASA Ames-Dryden. These are given at two span stations. The inboard span station (72 inches from the symmetry plane) corresponds to the location just inboard of the laminar flow control glove. Since the computational grid lines did not correspond to constant span stations, cubic spline interpolation was necessary to compute the flow variables at the appropriate span stations. The solid lines indicate the computations and the rectangles indicate experiment. For the inboard station, pressure taps were instrumented up to 25 percent of local chord, while outboard taps were instrumented up to 40 percent of local chord. The computations at the inboard station compare fairly well with the experimental data, with a slight underprediction. The slight underprediction occurs at 2-9 percent of chord. The computations are in excellent agreement with experiment from 10 percent of chord onward, and compare fairly well at the leading edge. At the outboard station, the computations consistently underpredict the experiment over the entire chord. However, there may be twist at this span station in the actual geometry which has not been accounted for in the computational model.

M = 1.6, $\alpha = 2.0^{\circ}$, Re_L = 116 million

COMPUTATIONS
TEST DATA (DRYDEN)



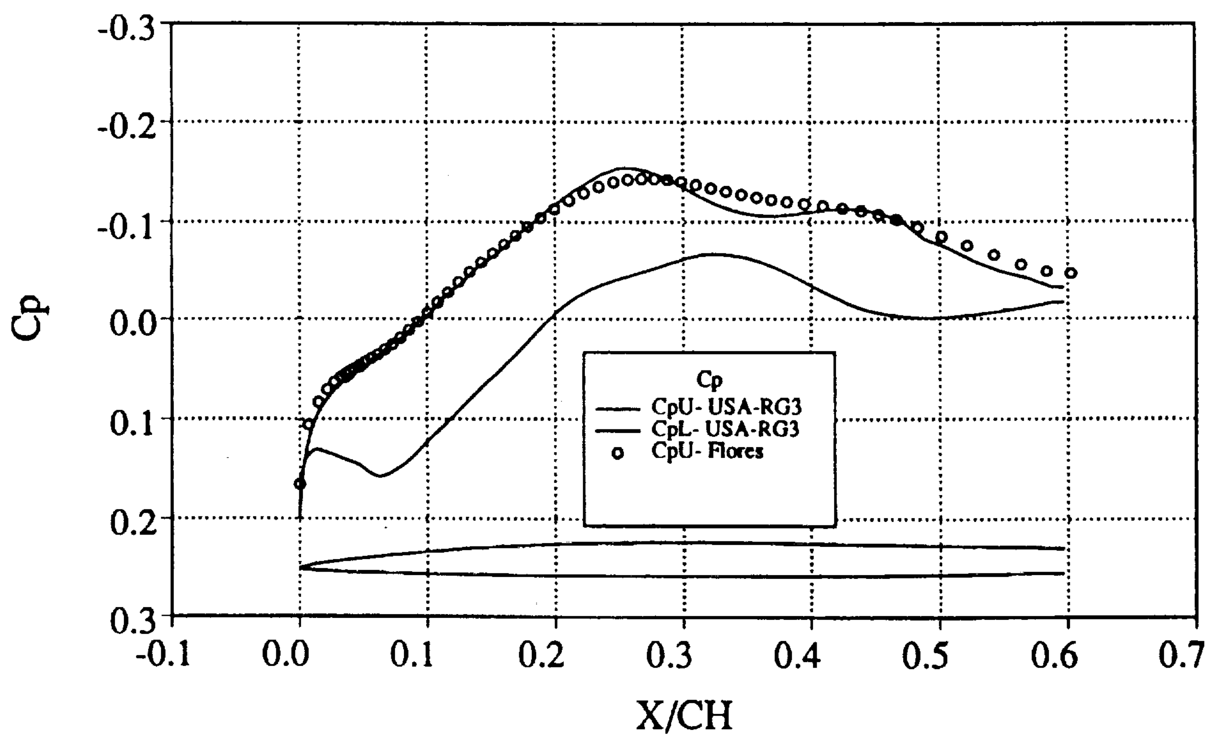
INBOARD STATION (72")



OUTBOARD STATION (114")

COMPUTATIONAL COMPARISONS OF PRESSURE COEFFICIENT

In the previous result, the computations underpredict the experimental pressure coefficients, especially between 2-9 percent of local chord at the inboard station. A comparison of the numerical results obtained by the CNS code and that due to a completely different code, called the USA-RG3 code[5] developed by Rockwell International, was conducted using the same grid. As can be noted, there is very good agreement between the two numerical results at the inboard station. In particular, where the CNS results were quite different from experiment in the 2-9 percent local chord region, there is excellent agreement obtained there between the two different codes.



VELOCITY PROFILES AT THE INBOARD STATION

An examination of the velocity profiles is conducted along the inboard station at different chordwise locations. The y-axis is the vertical height, in inches, above the geometry. The streamwise component of velocity is discussed first. The boundary layers all exhibit the standard expected profile. The boundary layer near the leading edge is very thin relative to the downstream profiles. At x/c = 3.2 percent the boundary-layer maintains a fairly constant thickness downstream to about x/c = 4.7 percent. The boundary-layer thickness near the leading edge is approximately .015 inches.

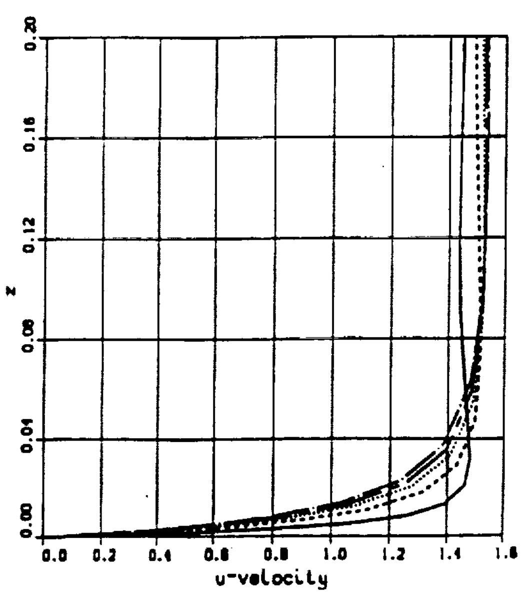
From examining the crossflow component it can be noted that the maximum crossflow occurs near the leading edge (x/c = .7 percent). At x/c = 2.2 percent, and downstream, the crossflow velocity has decreased dramatically. From x/c = 3.2 percent, and downstream, the crossflow velocity decreases continually, but not significantly. The inflection point of the crossflow velocity profile increases in height for the first three chordwise locations and then appears to decrease.

$M = 1.6, \ \alpha = 2.0^{\circ}$

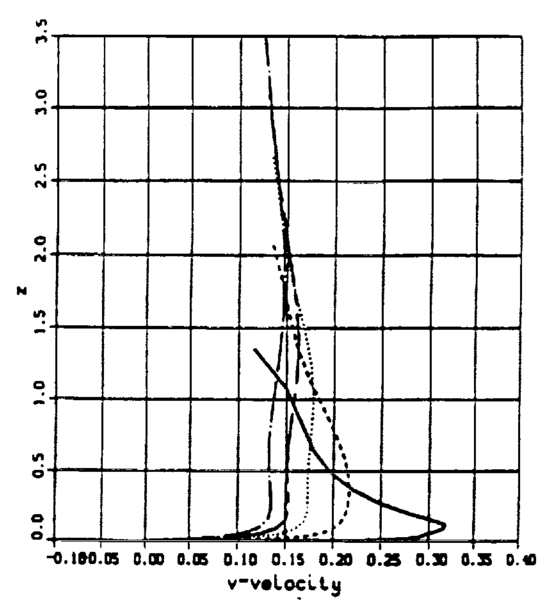
$Re_L = 100$ million

X/C

———	0.70 %
-----	2.20 %
········	3.20 %
—-—	4.00 %
—·—	4.70 %



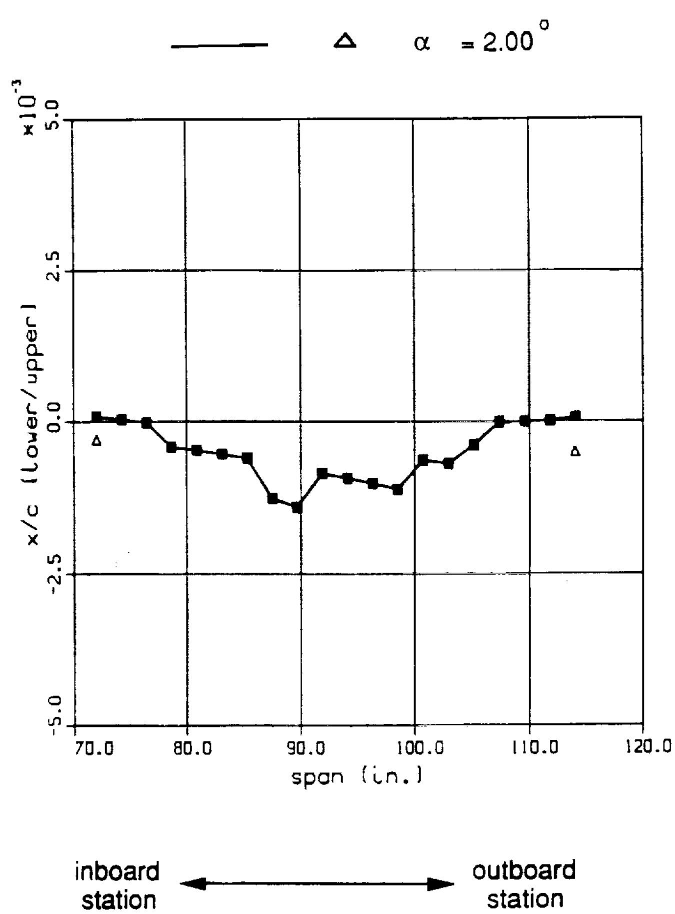
STREAMWISE VELOCITY



CROSSFLOW VELOCITY

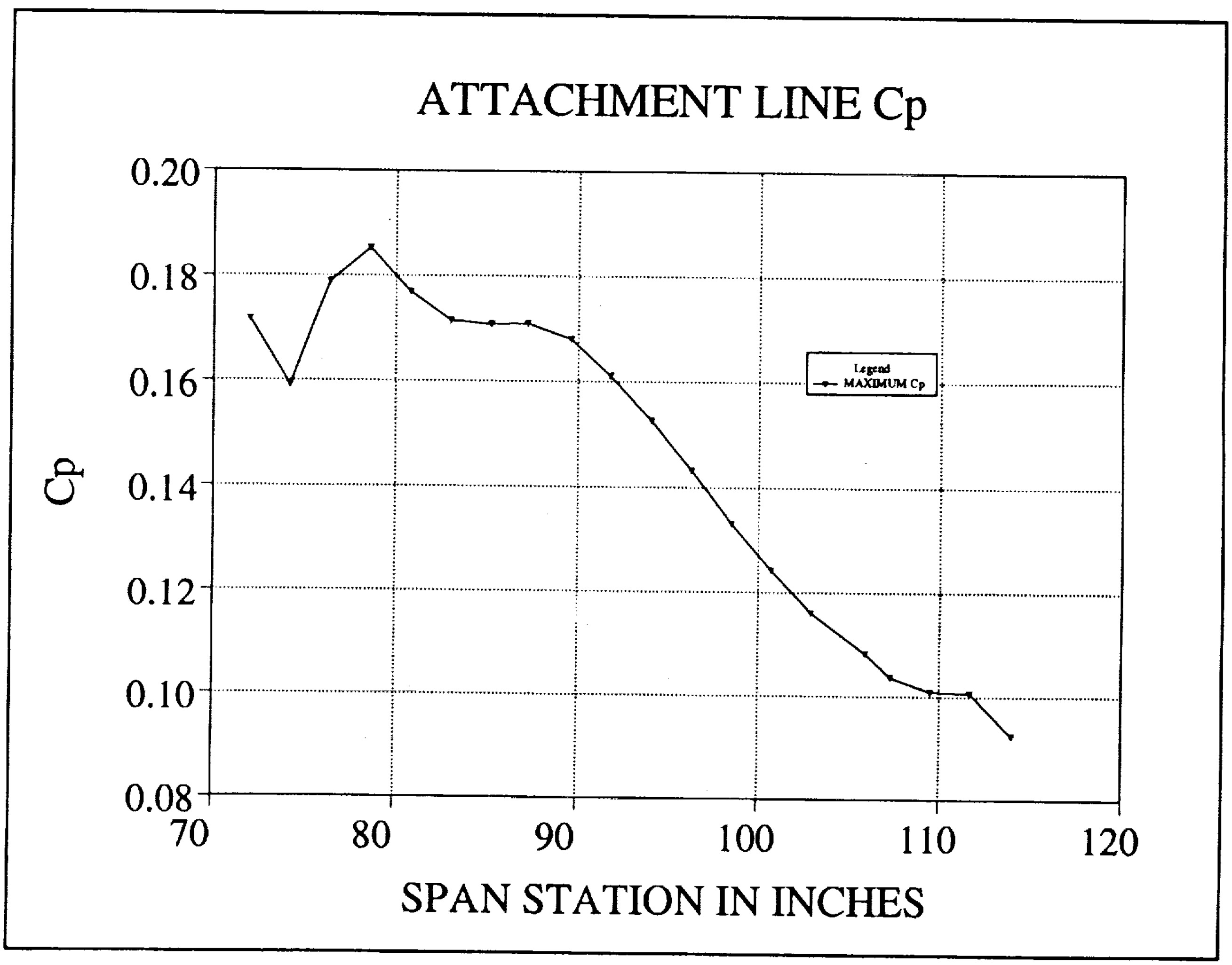
ATTACHMENT LINE LOCATIONS

The following results map the movement of the attachment line location from the inboard location of the wing to the outboard location. The experimental data points exist only at the inboard and outboard portion of the glove. The vertical axis indicates the position of the attachment point, either on the upper surface, (positive x/c) or on the lower surface (negative x/c). The leading edge itself is at $x/c = 0.0$. There are twenty equally-spaced interpolated span stations between the inboard and outboard stations. The stagnation point, at each span station, was determined by finding the grid point corresponding to the maximum pressure coefficient and is consistent with the experimental determination of the stagnation point. The procedure accounts for the discontinuities in the plot. The computations seem to indicate that the stagnation point is right on the leading edge of the inboard station, then goes below the leading edge at 75 inches and stays on the lower portion of the wing. At about 110 inches, the stagnation point returns to the leading edge of the wing. The experimental data points indicate the stagnation points slightly below the leading edge at both inboard and outboard stations. Other computational results[5] indicate the same trend as the current computational results.



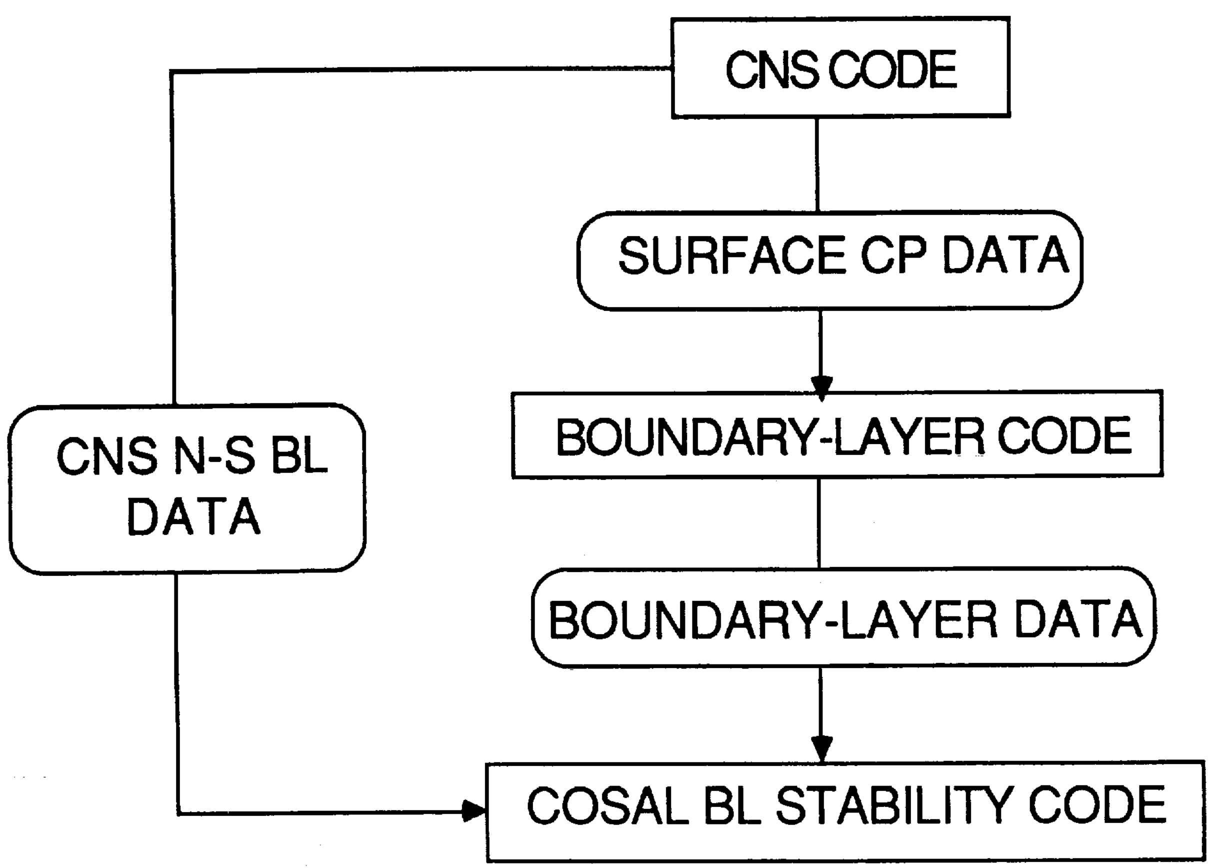
PRESSURE COEFFICIENT ALONG ATTACHMENT LINE

The following results reflect the pressure coefficient at the stagnation point from the inboard to outboard station. There is a dip in the pressure coefficient at a span station of about 74 inches. This location is about two inches away from where the inboard portion of the glove begins. The pressure coefficient then shows a favorable pressure gradient along the attachment line and levels off at a span station of about 110 inches. The last data point indicates that the pressure coefficient may take another dip here, which interestingly occurs close to the location where the glove ends. This result indicates that there may be some effect of how the glove is faired into the original wing.



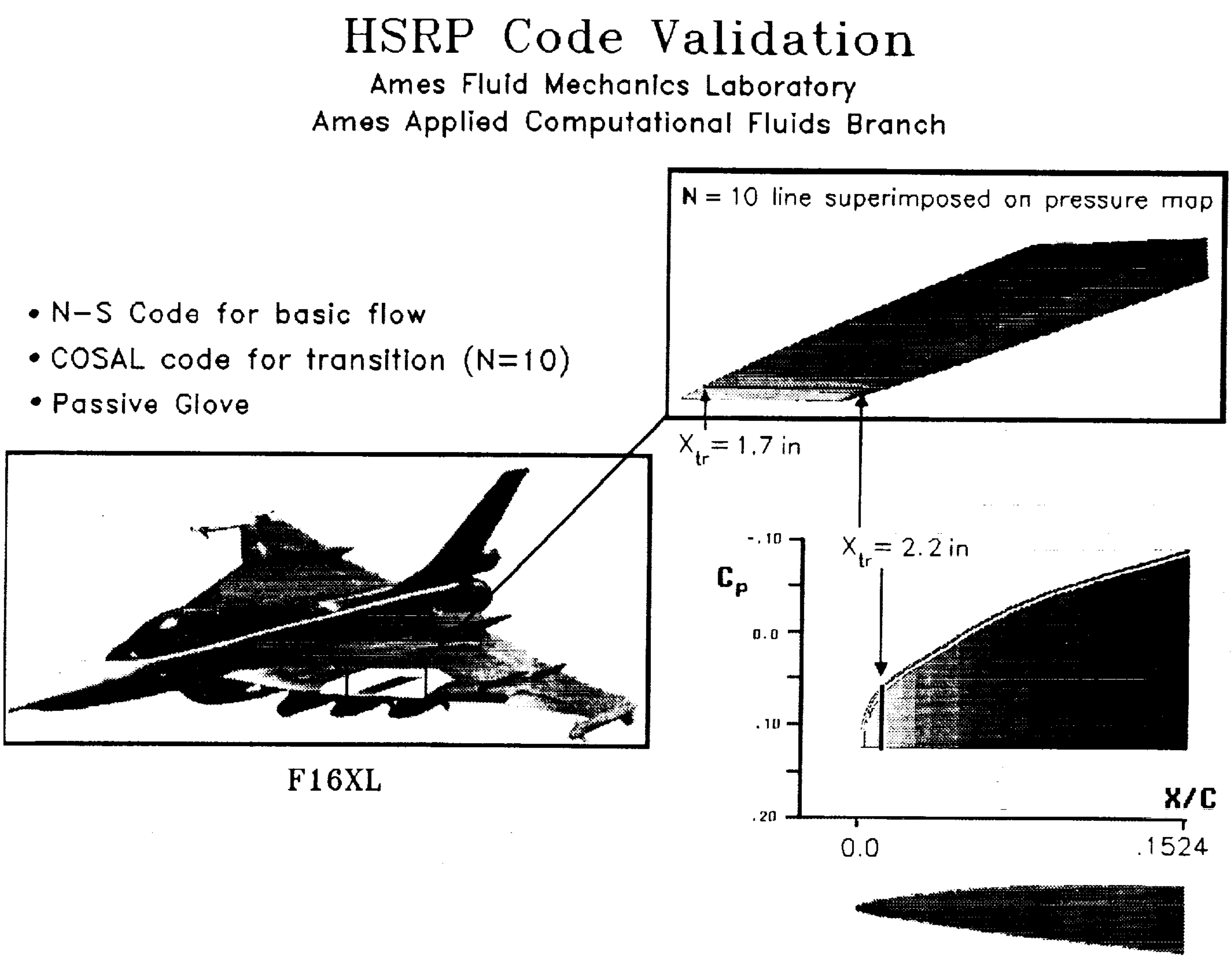
FLOW SOLVER - STABILITY CODE COUPLING

The Navier-Stokes code is currently coupled to the stability code in the following manner. The pressure distribution at various span stations from inboard to outboard is read into the boundary-layer code. The boundary-layer code uses a conical flow assumption in computing its boundary-layer data based on the given pressure distributions. This boundary-layer data is then read into the stability code. Depending on the N-factor value prescribed for the determination of transition, the stability code will determine the x/c location of transition for each span station. Future work will be performed to couple the Navier-Stokes solution from the CNS code directly into the stability code.



HSRP CODE VALIDATION

A composite result is displayed which illustrates the end product of the transition predicted by the CNS - COSAL code coupling. An N-factor equal to 10 was used in the COSAL code. The white area on the F-16XL wing designates the glove location. The box illustrates an expanded view of a small section of the glove. At a span station of 89 inches, transition occurs at about 1.7 inches (.6 percent of x/c) from the leading edge. Slightly outboard of that transition occurs at about 2.2 inches (.9 percent of x/c). It can be noted that for this case, transition occurs very close to the leading edge which is consistent with experimental findings. The corresponding C_p for the outboard location is also illustrated. The leading edge geometry of the wing is also indicated below the C_p graphic.



SUMMARY AND FUTURE DIRECTIONS

In summary, the CNS code has been used to predict the flow about the F-16XL in supersonic flight. Comparisons were made between the numerical and experimental pressure coefficients with good agreement between the two. Further numerical comparisons were conducted with the results from another Navier-Stokes code. Velocity profiles, for both streamwise and crossflow components, were analyzed at the inboard station for various x/c values. A mapping of the attachment line from the inboard to the outboard area of the glove was conducted. Finally, the numerical results from the CNS code were used in the COSAL code to predict transition.

SUMMARY

COMPUTED NUMERICAL SOLUTION OF THE FLOW FIELD ABOUT THE F-16XL IN SUPERSONIC FLIGHT

- -COMPARISONS OF THE NUMERICAL SOLUTION WITH IN-FLIGHT EXPERIMENTAL DATA WAS CONDUCTED
- -VELOCITY PROFILES FOR INBOARD STATION ANALYZED
- -MAPPING OF ATTACHMENT LINE LOCATION WAS CONDUCTED

CNS CODE COUPLED TO COSAL CODE

- -TRANSITION PREDICTED ON THE GLOVE PORTION OF THE WING

FUTURE DIRECTIONS

CNS CODE

- -CONTINUE VALIDATION OF THE CODE
- -IMPLEMENT SUCTION BOUNDARY CONDITIONS

CNS/COSAL CODE

- -MAP TRANSITION LINE AND VALIDATE WITH IN-FLIGHT DATA
- -ADD CAPABILITY TO UTILIZE NAVIER-STOKES SOLUTION DIRECTLY

REFERENCES

1. Malik, M.R., "COSAL - A Black Box Compressible Stability Analysis Code for Transition Prediction in Three-Dimensional Boundary-Layers," NASA CR-165925, 1982.

2. Mack, L.M., "On The Stability of the Boundary Layer on a Transonic Swept Wing," AIAA Paper No. 79-0264.

3. Mack, L.M., "Transition Prediction And Linear Stability Theory in Laminar-Turbulent Transition," AGARD Conference Proceedings No. 224, pp. 1-1 to 1-22, 1977.

4. Flores, J. and Chaderjian, N.M., "Zonal Navier-Stokes Methodology for Flow Simulation About A Complete Aircraft," Journal of Aircraft, Vol. 27, No.7, July 1990, pp. 583-590.

5. Woan, C.J., Gingrich, P.B., and George, M.W., "CFD Validation of a Supersonic Laminar Flow Control Concept," AIAA Paper No. 91-0188.