N76-28171

UTILIZATION OF THE AEDC THREE-DIMENSIONAL POTENTIAL FLOW COMPUTER PROGRAM*

Richard L. Palko
ARO, Inc.

SUMMARY

A potential flow computer program has been in use at the Arnold Engineering Development Center (AEDC) for several years. This program has been used primarily as a tool for flow-field analysis in support of test activities in the transonic wind tunnels of the Propulsion Wind Tunnel Facility (PWT). Analyses have been made over a Mach number range from 0 to 0.9 for a variety of configurations from aircraft to wind tunnels, with excellent agreement between calculated flow fields and measured wind tunnel data. Analytical and experimental data for seven different flow analysis problems are presented in this paper.

INTRODUCTION

The AEDC Three-Dimensional Potential Flow Computer Program (PFP) in the existing form was developed primarily as a result of the need to make calculations of the flow field in the vicinity of aircraft fuselages (typically at locations where aircraft inlets might be located). This need arose because of the support the theoretical flow-field calculations could lend to a research program carried out at AEDC to simulate the inlet flow fields in a wind-tunnel test of full/scale inlet/engine systems (refs. 1 and 2). Much of the computing capability that the PFP presently has resulted from these flow-field calculations which have as their primary variables the flow angularity (upwash and sidewash) over a y-z plane. After the initial solution of the velocity field for a given model attitude and Mach number is obtained, the upwash and sidewash can be determined for any given point or over any grid desired. A new solution is required for each model attitude or Mach number. In addition to computing the upwash and sidewash, the PFP also computes the local Mach number, C_p, and flow streamlines. A computer plotting program has been written to supplement the PFP, and computer plots can be

*The research reported herein was conducted by the Arnold Engineering Development Center, Air Force Systems Command. Research results were obtained by personnel of ARO, Inc., Contract Operator at AEDC. Further reproduction is authorized to satisfy needs of the U. S. Government.

8

obtained for most of the above parameters. A streamline can be traced from any point in the stream either upstream or downstream (or both). The capability to make a plot of the mathematical representation of the model geometry before running the complete program allows corrections to be made, if needed, with only a slight loss of computer time. A two-volume report (ref. 3) that details the program, modeling techniques, application, and verification has been published.

This paper briefly outlines the PFP application to seven flow analysis problems in support of the transonic wind tunnels in the PWT at AEDC.

SYMBOLS

Values are given in both SI and U. S. Customary Units. The measurements and calculations were made in U. S. Customary Units.

A	angle of attack
C_L	lift coefficient
C_p	pressure coefficient
M_∞	free-stream Mach number
p/p_o	ratio of surface static pressure to free stream total pressure
X	coordinate along tunnel axis, positive downstream
Y	horizontal coordinate, sign as indicated
Z	vertical coordinate, sign as indicated
α	model angle of attack, deg, positive up
β	model angle of yaw, deg, sign as indicated
ϵ	upwash, deg, positive up
σ	sidewash, deg, positive as indicated

PFP APPLICATION AND UTILIZATION

The PFP at AEDC has been used primarily as a tool for analysis of the flow in the far field. (Far field refers to a distance away from the analysis model surface equal to, or greater than, the vortex spacing in the direction of flow.) The modeling techniques required for this type of analysis are presented in volume II of reference 3. However, work is underway to develop the modeling technique to allow accurate analysis of the surface pressure. Results of some of this continuing effort are reported in reference 4. All the flow problems presented here are of the far-field type.

Flow Field Between Two Hollow Circular Cylinders

The analysis of these cylinders was part of a research program in which the objective was to create flow fields by some auxiliary method to simulate the flow entering a full-scale inlet/engine at high angles of attack and yaw. The device was to deflect (or induce) the flow upward as it passed between the inclined cylinders. A mathematical model of the cylinders is shown in figure 1. The last circumferential ray on each cylinder had trailing vortices that were trailed at an angle equal to one-half the cylinder pitch angle. A comparison between the theoretical and experimental flow angularity data is shown in figure 2. The theoretical results are shown as lines of constant flow angle (both upwash and sidewash), and the solid symbols show the relative location of experimental data with the magnitude of the measured angles indicated. The Mach number at which these data were taken was 0.9. It can be seen that the PFP overestimated the flow inclination angles by approximately 1°.

Flow Field Around an Aircraft Fuselage

Primary purpose of this analysis was to verify the results obtained from the PFP. Experimental flow-field data used for comparison with theory were available from wind-tunnel measurements made during the Tailor Mate test series. The objective of the wind-tunnel test was to determine the flow field (upwash and sidewash) at a typical engine inlet fuselage location. The mathematical model of the fuselage configuration is shown in figure 3. The comparison between the predictions from the PFP and the wind-tunnel data for a pitch angle of 25° and a free-stream Mach number of 0.9 are shown in figure 4. Here again, excellent agreement was obtained.

Flow Field Under a Fuselage-Wing Configuration

The purpose of this analysis was also for program verification; again experimental data obtained during the Tailor Mate studies were used. The fuselage-wing configuration was analyzed to compute the flow field under the wing at the wing-fuselage junction. The computer math model used in the analysis is shown in figure 5. A comparison between the upwash and sidewash predictions and the experimental data for a Mach number of 0.9 and an angle of attack of 10° is shown in figure 6. Analytical and experimental data trends show excellent agreement, although the predicted data show somewhat higher flow angularity gradients across the survey area than the measured data.

Streamtube Entering Inlet Behind Wing

The purpose of this analysis was to determine the origin of the streamtube entering the inlet in support of an inlet hot gas ingestion investigation. The comparison between experimental and theoretical data are shown in figure 7. These data were taken during a store separation study in an effort to verify the accuracy of the PFP to predict the correct flow field above and behind the wing. Data were taken at a Mach number of 0.3 with an angle of attack of 8°, and show excellent agreement between the experimental and theoretical values. The mathematical model and the predicted streamtube are shown in figure 8. The streamtube was determined by tracing streamlines from four locations beginning just upstream of the inlet and extending forward to just upstream of the aircraft nose. A Mach number of 0.3 and an angle of attack of 8° were also used for the streamtube analysis. This mathematical model is the largest analyzed to date, with 1559 loop vortices and 20 horseshoe vortices, and required approximately 4 hours run time on the AEDC IBM 370/165 computer.

Inlet/Engine in Crosswind

This analysis was made in support of a crosswind experiment conducted during an inlet study in the AEDC 16-ft (4.88-m) Transonic Wind Tunnel (PWT-16T). The objective of the analysis was to determine if a 0.91-m-diameter (3-ft-diameter) crosswind simulator would adequately simulate the crosswind when used in conjunction with the inlet model, and to determine the position for the simulator to give best results. The theoretical analysis was made with the inlet/engine in an infinite crosswind. The mathematical model included only a portion of the experimental model as shown in figure 9. A computer plot of the mathematical model is shown in figure 10. The engine ducts were closed on the downstream end and a negative source was located near the rear center of each engine duct to produce the correct inlet mass flow when that particular engine was in operation. Streamlines were traced upstream from near the four corners of the inlet, for each engine in operation, to determine the flow pattern of the airstream entering the inlet. By tracing the streamlines, a fan position was determined that would influence the inlet flow for all engine power settings and crosswind velocities required. A typical flow pattern for the analysis is shown in figure 11 for a crosswind velocity of 20.57 m/sec (67.5 ft/sec) with both engines operating.

Pressure Distribution in PWT-16T Contraction Section

The objective of this analysis was to determine the pressure distribution along the bottom and side walls of the PWT-16T contraction section. Pressure distributions were needed for use in a theoretical boundary-layer analysis of the wind-tunnel nozzle to support a test-section flow angularity study. Mathematical modeling used in the analysis is shown in figure 12. The flow in the test section area was specified to give the pressure ratio desired for Mach number 0.6. The analysis provided streamline information at a distance of 0.305 m (1 ft) from the walls, and the calculated C_p was converted to p/p_o. Following the calculation of the theoretical pressure distribution, the pressure distribution was experimentally measured in the contraction section. A comparison between the theoretical and experimental pressure distribution is shown in figure 13, with excellent agreement indicated.

Strut Effects Analysis

The objective of this analysis was to determine the strut effect corrections to measured force and moment data for a slender winged vehicle with a mid-strut mount. The vehicle wing was located just forward of the strut. For this analysis the upwash angle was determined with the PFP for the body alone (fig. 14) and the body with strut (fig. 15). An incremental upwash angle was then determined at the wing location from these two sets of data. In this case the incremental values were negative because of the down flow around the strut. The incremental values along the wing location were averaged and the ΔC_L correction calculated from the average angle-of-attack change. A comparison of the calculated corrections and those measured with a subscale model are shown in figure 16. Excellent agreement is shown in both the trend with Mach number and the absolute values.

CONCLUDING REMARKS

The AEDC Potential Flow Program is used primarily as a tool for flow-field analysis in support of the test activities in the transonic wind tunnels of PWT. This paper has covered seven different problems that have utilized the PFP including both external and internal analysis. All but one of the examples have experimental data to verify the calculated flow fields, and all comparisons show excellent agreement. The PFP at AEDC has not been used as a tool to obtain absolute values, but rather as a tool to predict and verify flow fields in support of the test activities. In addition to the problems presented, the PFP has been used to predict the flow angularity at the model resulting from sting and support systems, to predict the flow around various types of support systems, and many other general flow analysis problems directly connected with wind-tunnel testing.

REFERENCES

1. Palko, R. L.: Full-Scale Inlet/Engine Testing at High Maneuvering Angles at Transonic Velocities. AIAA Paper No. 72-1026, Presented at the AIAA Seventh Aerodynamic Testing Conference, Palo Alto, California, September 13-15, 1972.

2. Palko, R. L.: A Method to Increase the Full-Scale Inlet/Engine System Testing Capability of the AEDC 16-Ft Transonic Wind Tunnel. AEDC-TR-73-9 (AD 762912), June 1973.

3. Todd, Donald C. and Palko, Richard L.: The AEDC Three-Dimensional, Potential Flow Computer Program. Vol. I and II. AEDC-TR 75-75, February 1976.

4. Heltsley, Fred L.: Report on the Status of a Slotted Wind-Tunnel Wall Representation Using the Vortex-Lattice Technique. Vortex-Lattice Utilization, NASA SP-405, 1976. (Paper no. 9 of this compilation.)

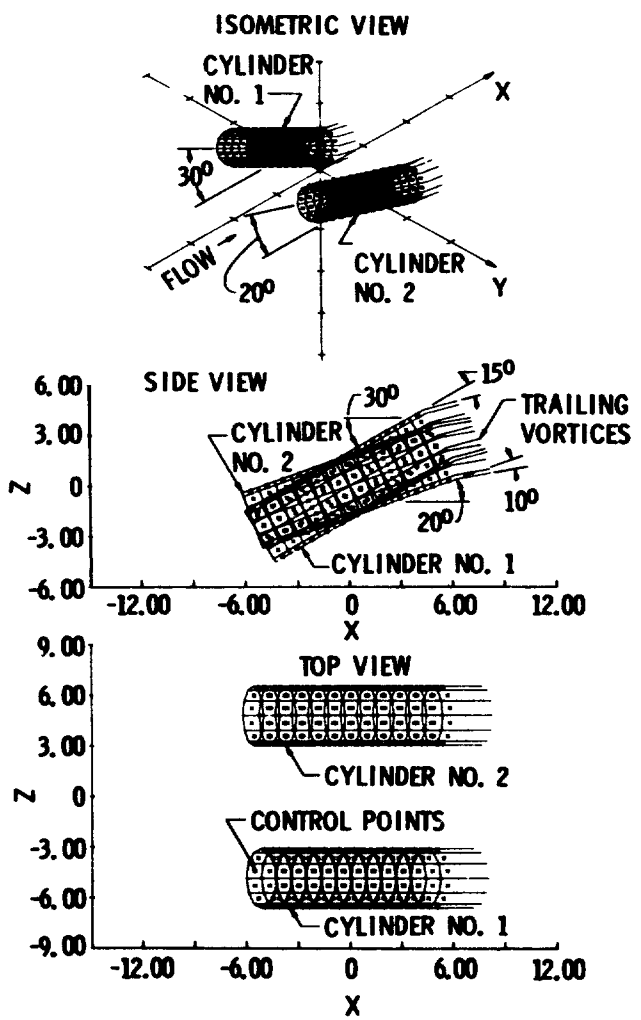

Figure 1.- Mathematical model of the dual hollow circular cylinder configuration.

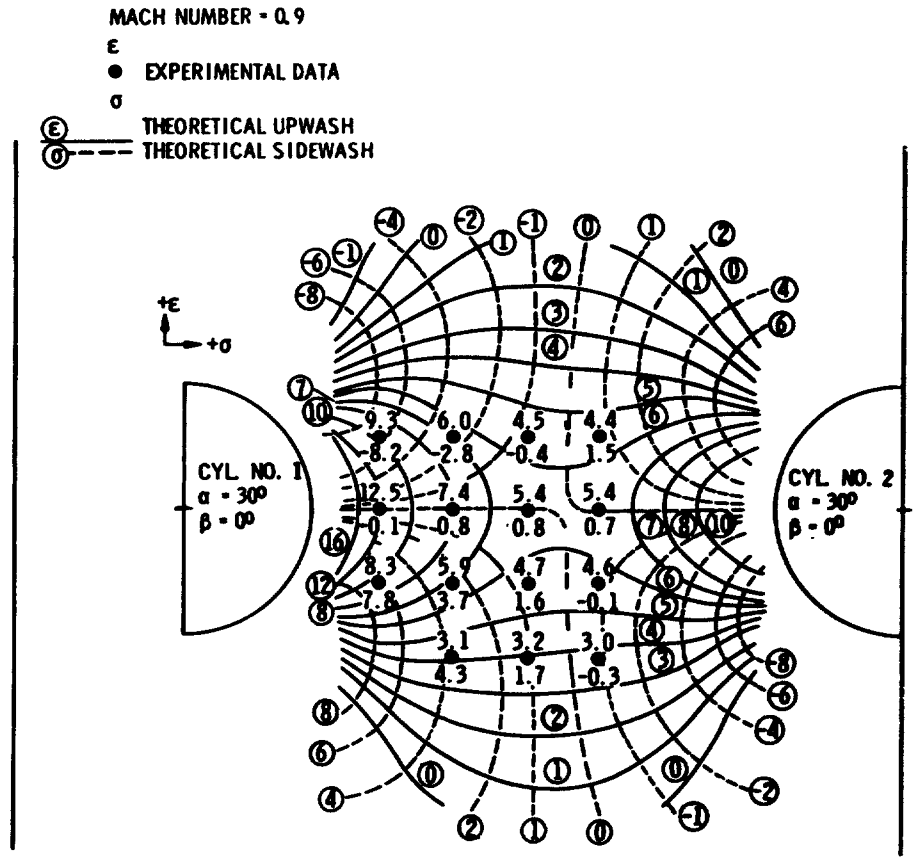

Figure 2.- Experimental and theoretical comparison of upwash and sidewash angles for the dual hollow circular cylinders at a Mach number of 0.9.

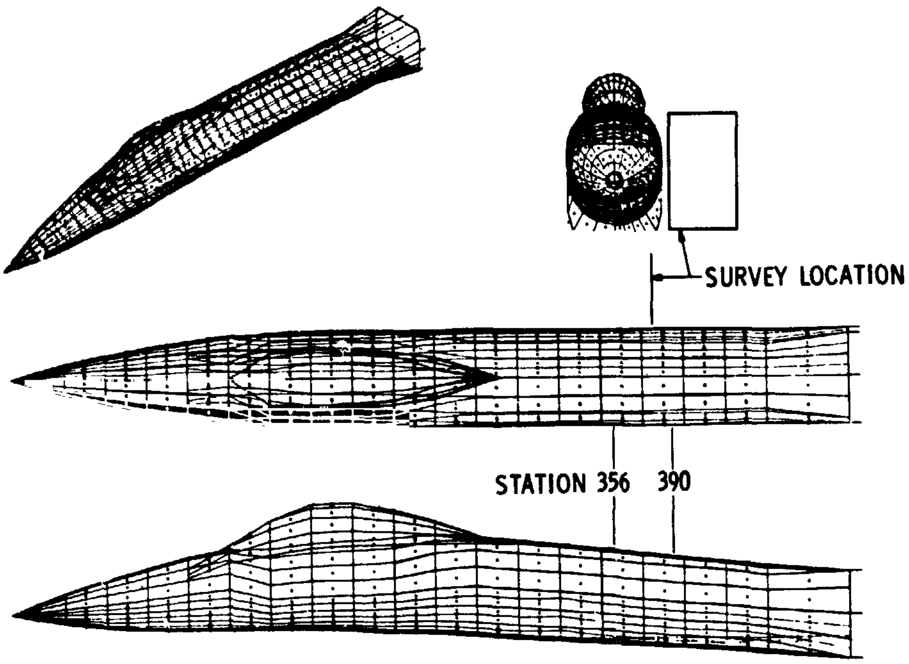

Figure 3.- Mathematical model of the fuselage configuration.

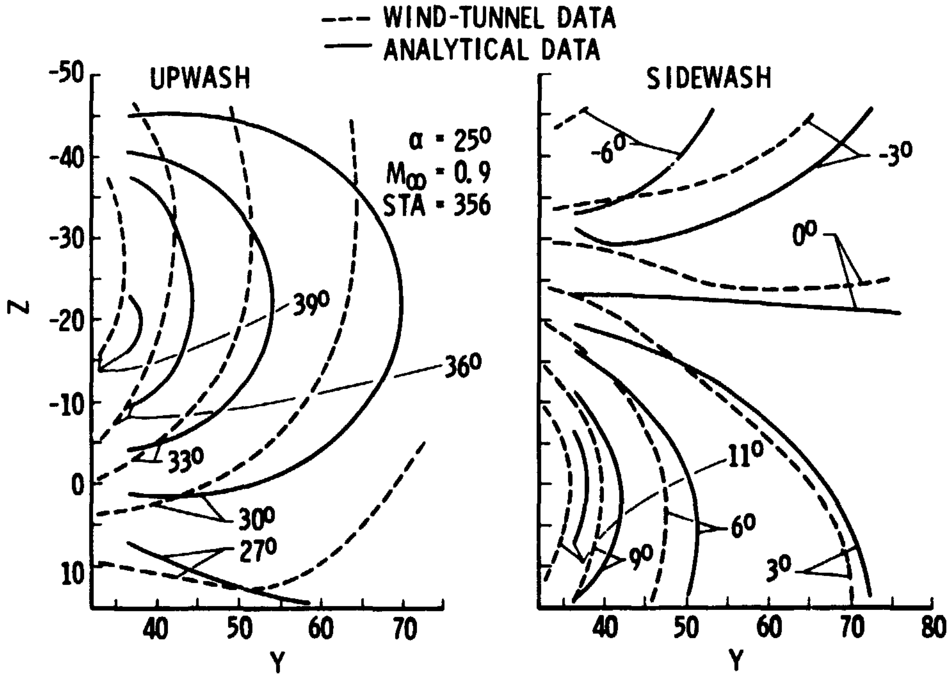

Figure 4.- Comparison between analytical and wind-tunnel data for Mach number 0.9 at an angle of attack of 25°.

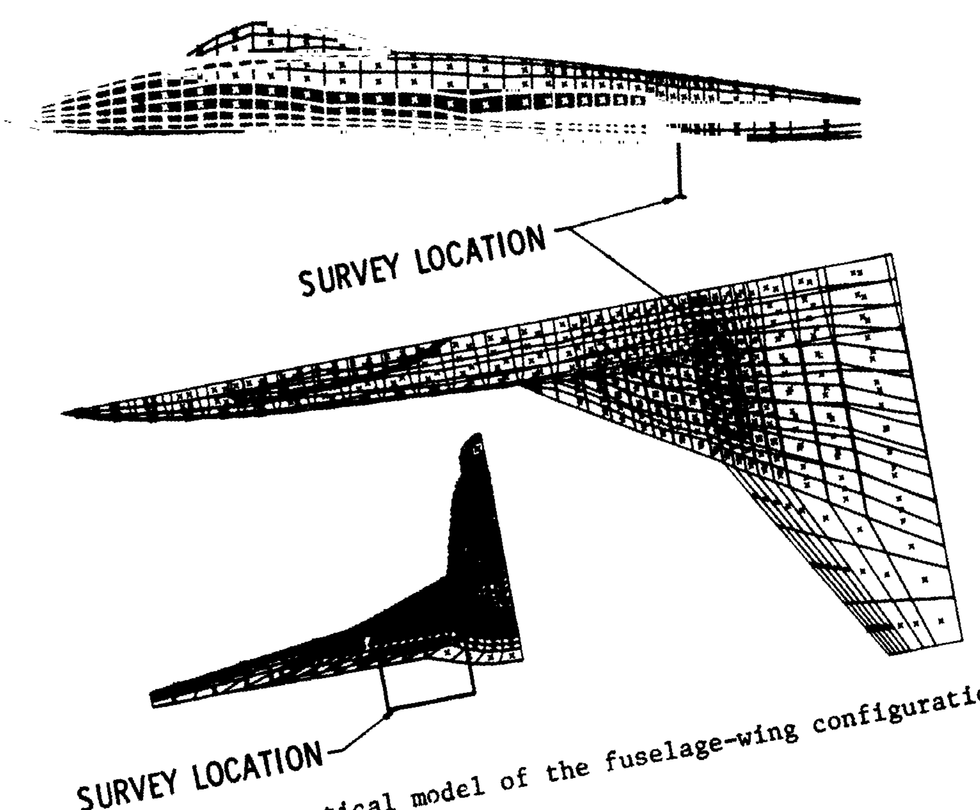

Figure 5.- Mathematical model of the fuselage-wing configuration.

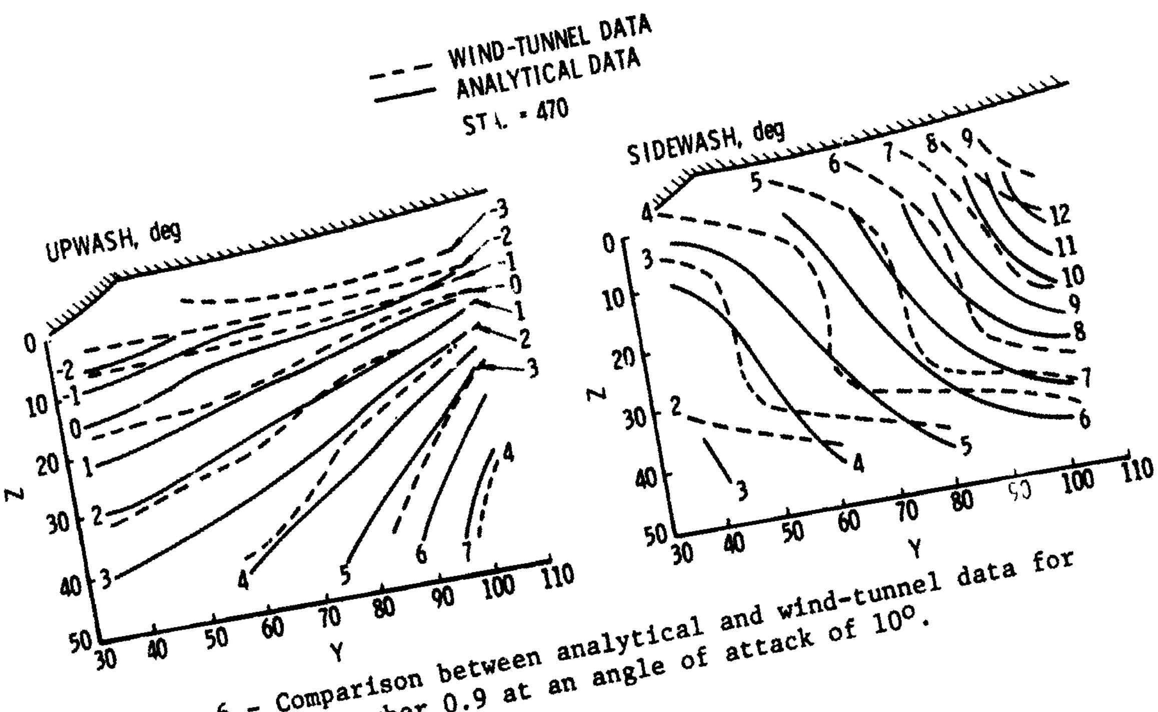

Figure 6.- Comparison between analytical and wind-tunnel data for Mach number 0.9 at an angle of attack of 10°.

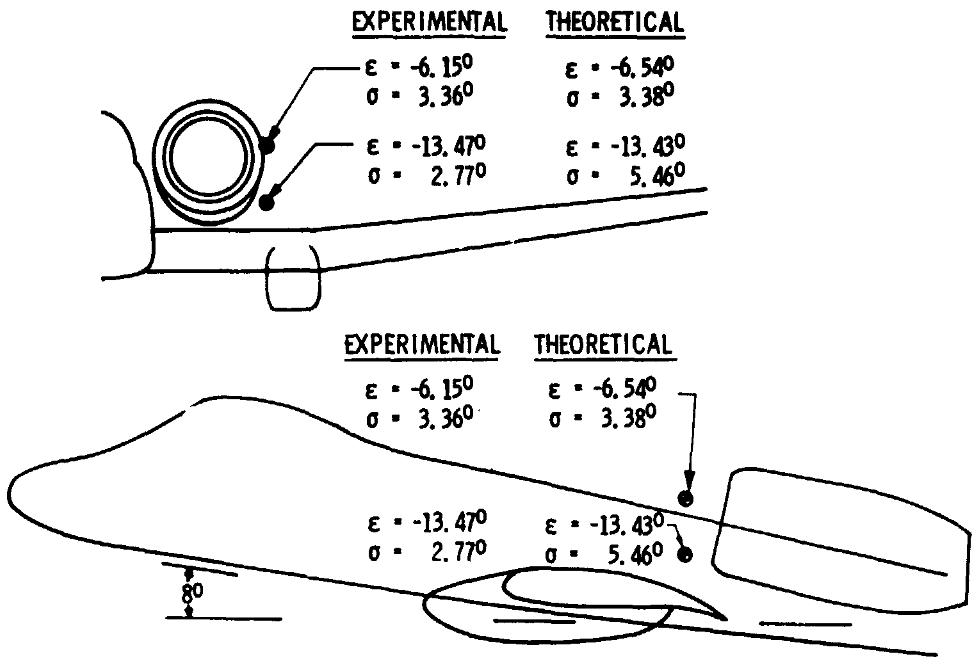

Figure 7.- Comparison between experimental and theoretical data for Mach number 0.3 at an angle of attack of 8°.

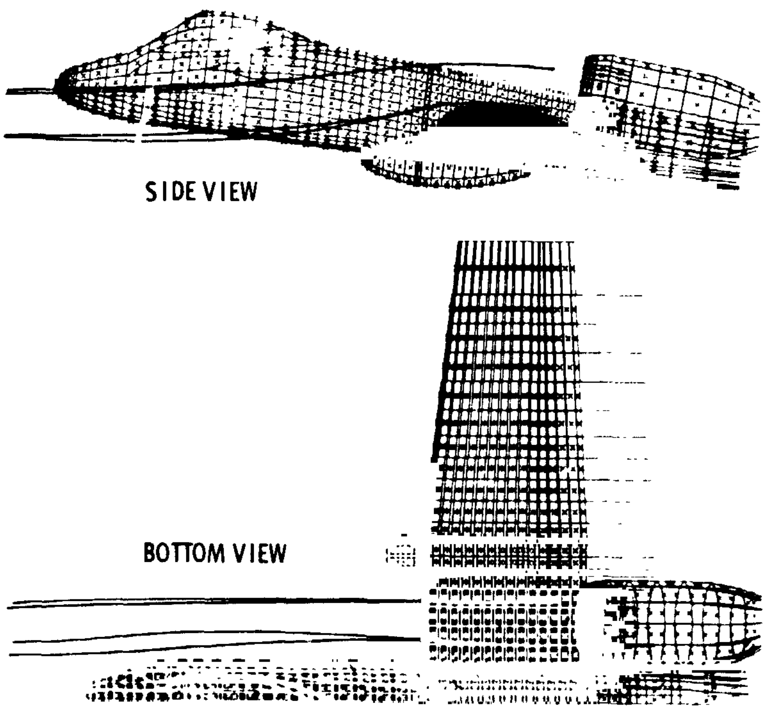

Figure 8.- Flow streamtube entering the inlet at Mach number 0.3 at an angle of attack of 8°.

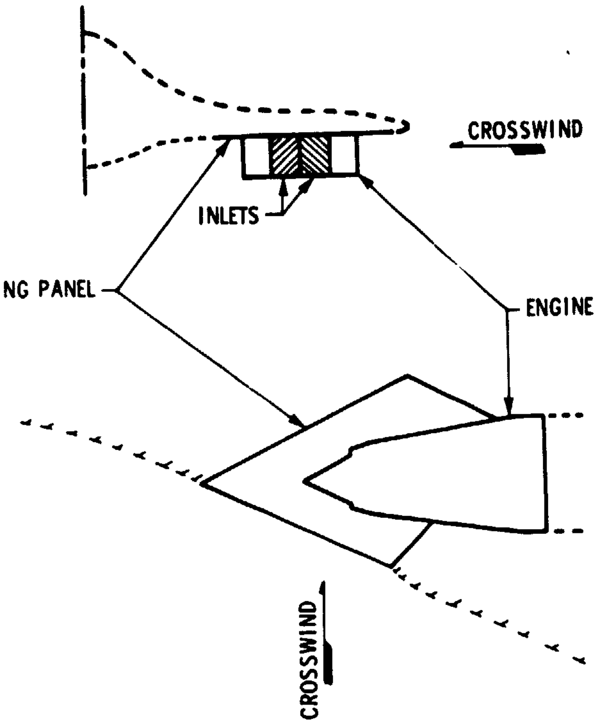

Figure 9.- Sketch of the section of the test model duplicated with the mathematical model.

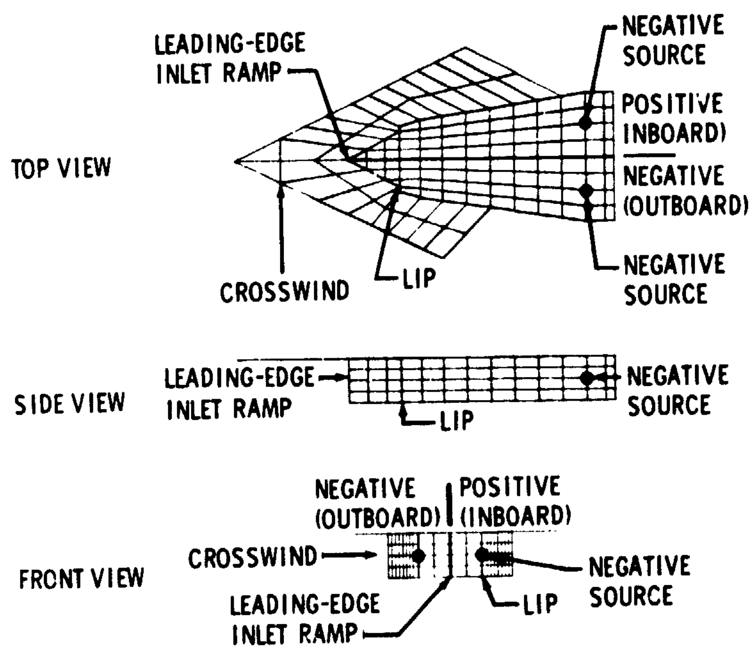

Figure 10.- Basic mathematical model for the inlet/engine crosswind analysis.

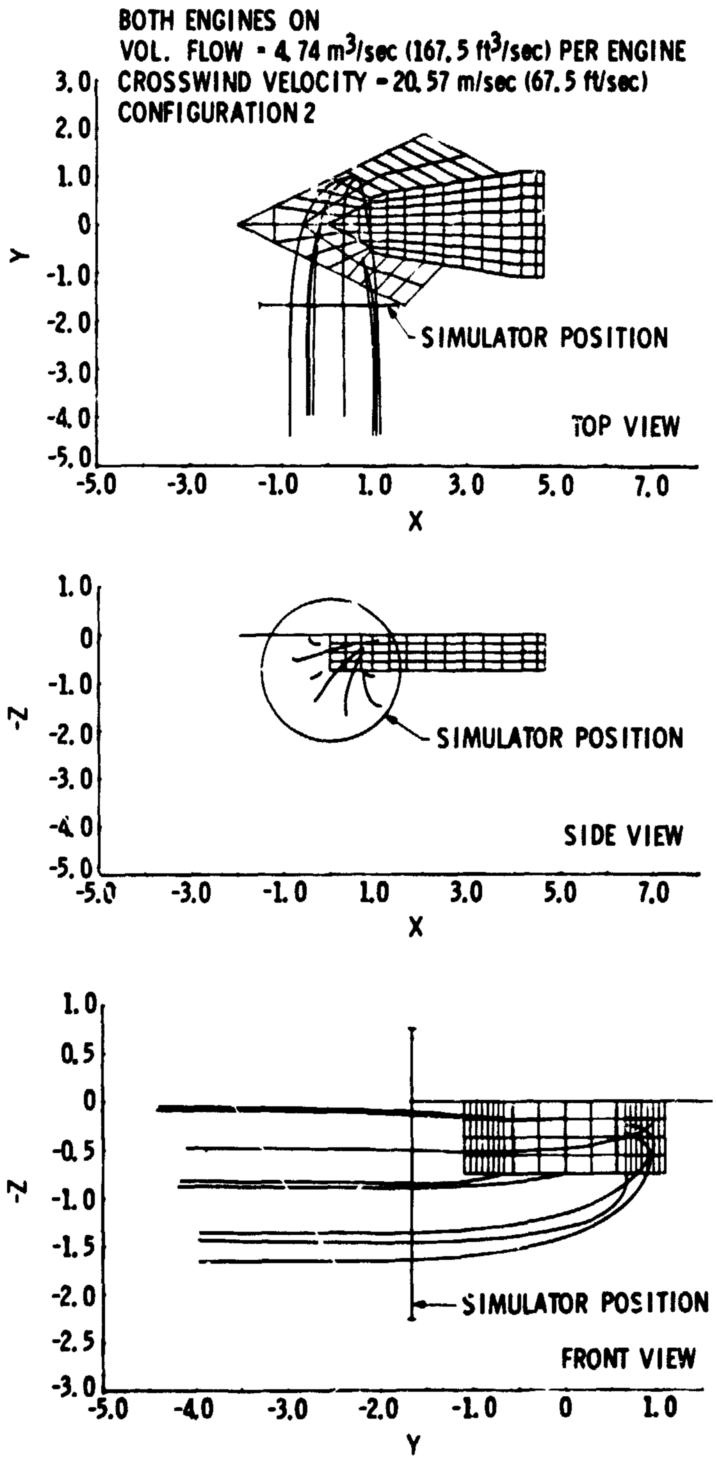

Figure 11.- Flow streamlines for both engines on with a crosswind velocity of 20.57 m/sec (67.5 ft/sec).

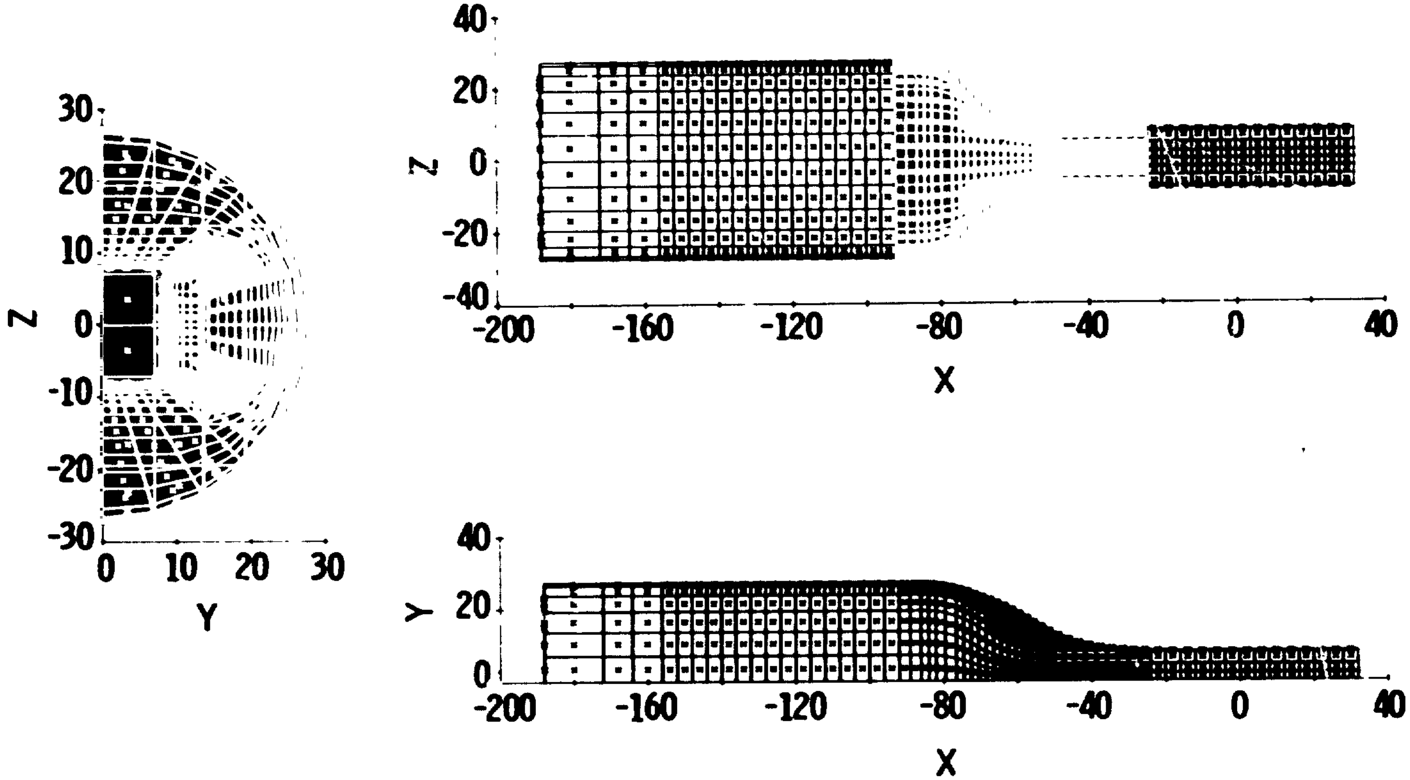

Figure 12.- Mathematical model of the PWT-16T ontraction section.

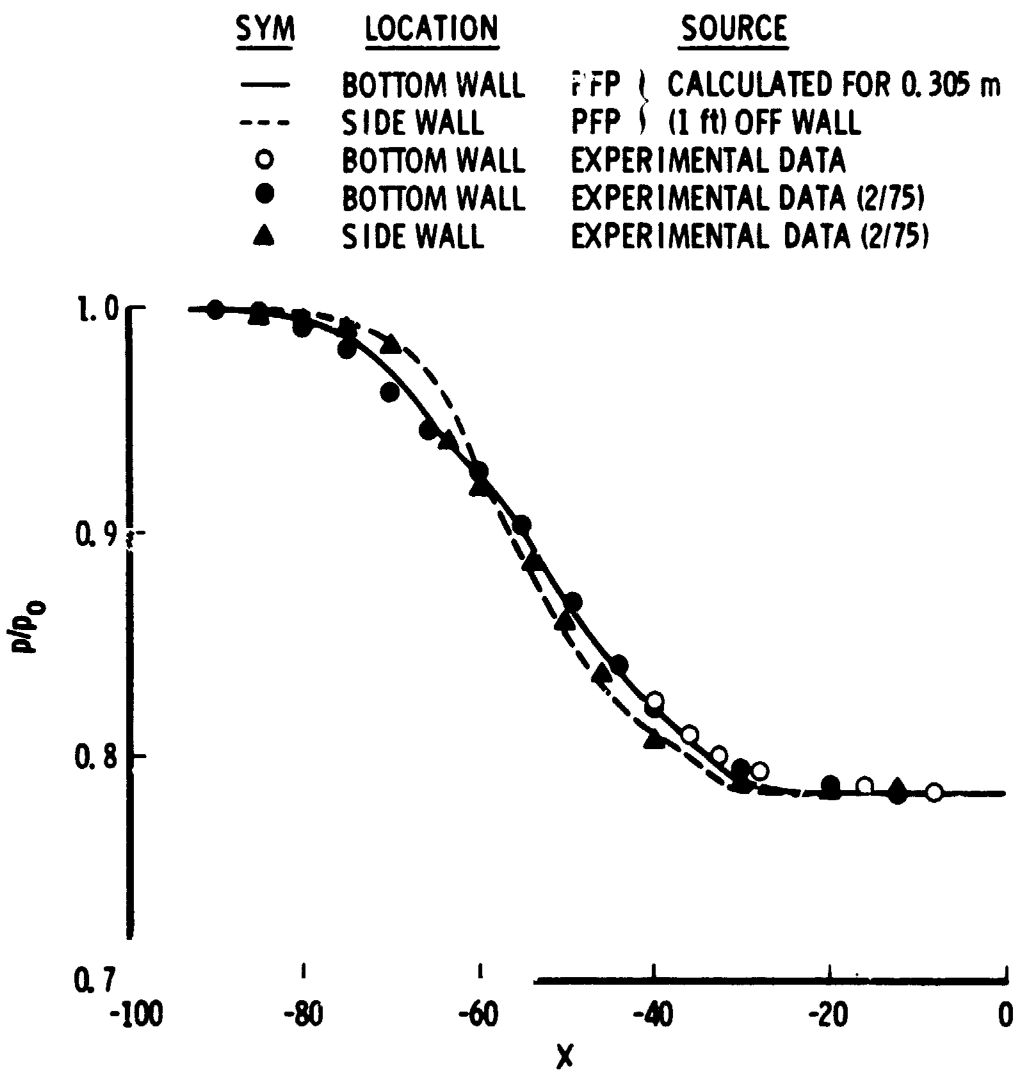

Figure 13.- Pressure distribution in PWT-16T for Mach number of 0.6.

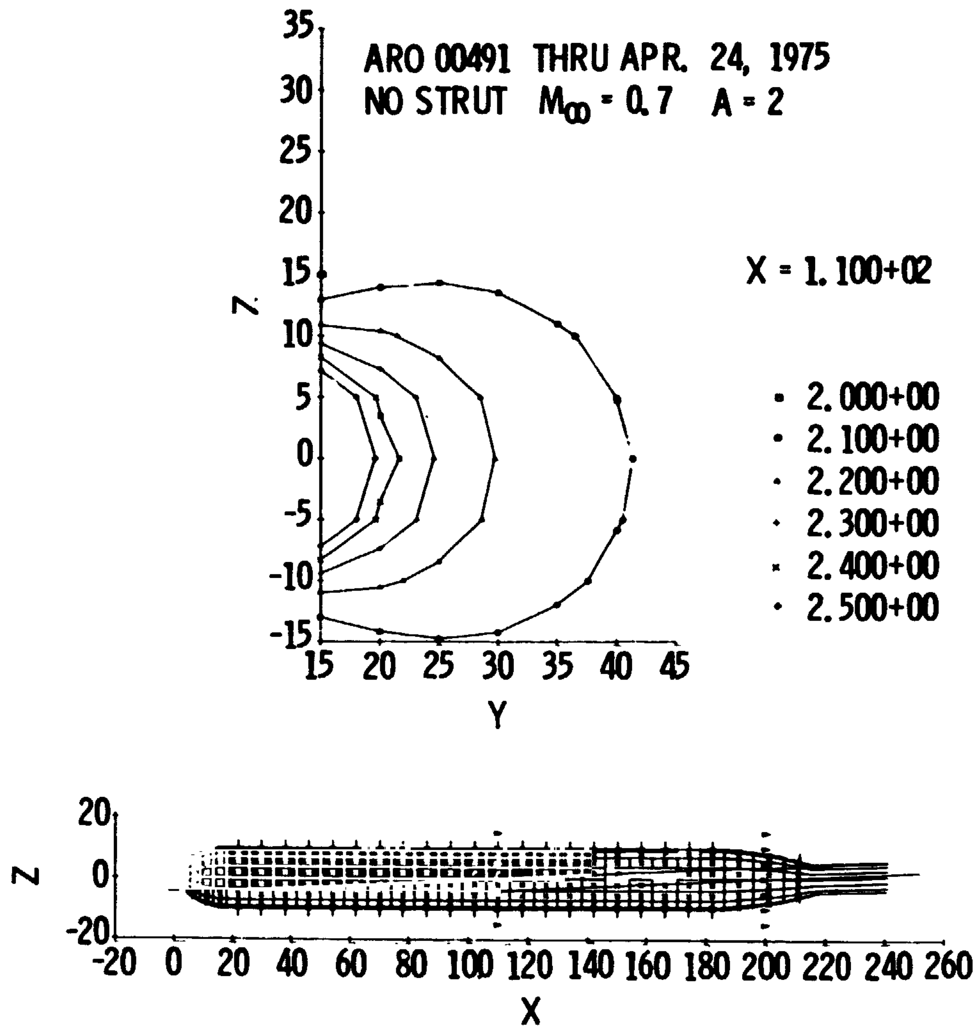

Figure 14.- Upwash about cylindrical model at $M_\infty = 0.7$ and $\alpha = 2^o$.

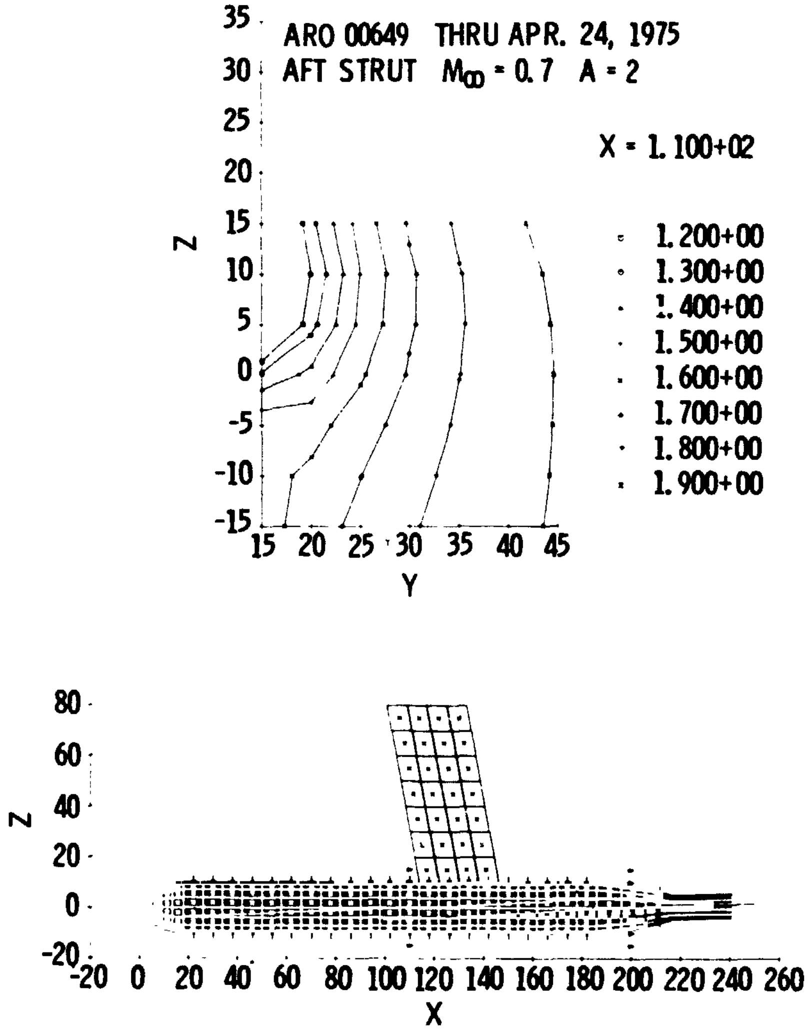

Figure 15.- Upwash about cylindrical model with strut at M_∞ = 0.7 and α = 2°.

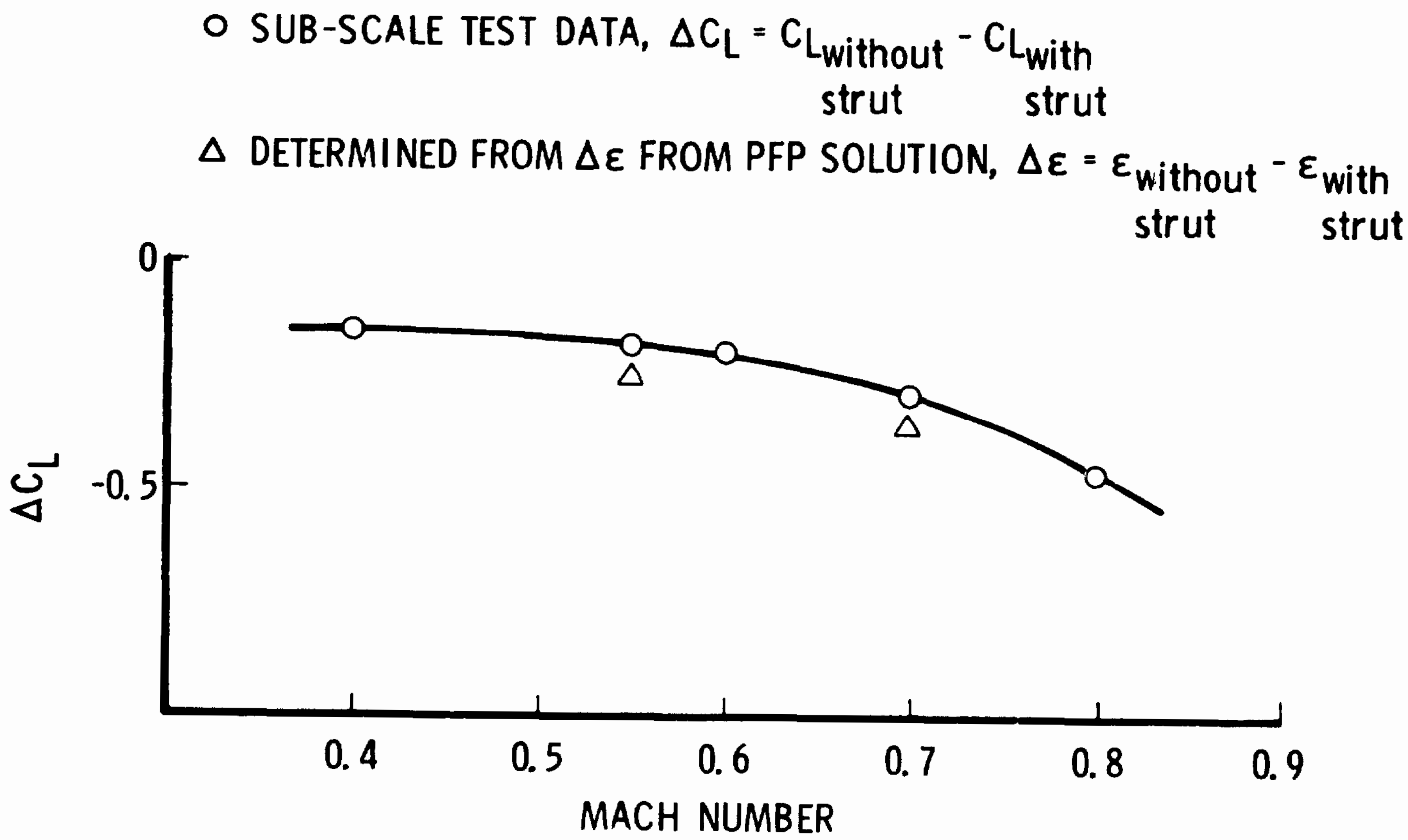

Figure 16.- Verification of strut effects of data.