California Polytechnic State University
San Luis Obispo, CA 93407

Computational Investigation of Flap-Edges

Final Report
April 1997

NASA Award NCC-2-857

Russell M. Cummings
Aeronautical Engineering Department

COMPUTATIONAL INVESTIGATION OF A PART-SPAN SLAT

Sandra L. Grabhorn
Aeronautical Engineering Department
California Polytechnic State University, San Luis Obispo, CA 93407

Russell M. Cummings
Aeronautical Engineering Department
California Polytechnic State University, San Luis Obispo, CA 93407

ABSTRACT

The current study expands the application of computational fluid dynamics to three-dimensional multi-element high-lift systems by investigating the flow dynamics created by a slat edge. Flow is computed over a three-element high-lift configuration using an incompressible Navier-Stokes solver with structured, overset grids processed assuming full turbulence with the one-equation Baldwin-Barth turbulence model. The geometry consists of an unswept wing, which spans the wind tunnel test section, a single element half-span Fowler flap, and a three-quarter span slat. Results are presented : the wing configured for landing with a chord based Reynolds number of 3.7 million. Results for the three-quarter span slat case are compared to the full-span slat and two-dimensional investigations.

INTRODUCTION:

High-lift aerodynamics continues to be and important area of research due to the powerful effect high-lift systems have on the overall design of an aircraft In order to continue the advancement of the performance of high-lift systems, a better understanding of the fundamental flow physics is needed. A number of authors have identified areas important to high-lift flows, such as viscous wake interaction, confluent wakes and boundary layers, and separated flows. Computational Fluid Dynamics (CFD) approaches have demonstrated an ability to capture many of these effects in two dimensions (Refs. 1, 2). This ability, coupled with the detailed flow information provided by a CFD solution, suggested a computational approach for this work. This study utilizes an incompressible Navier-Stokes solver in an attempt to further the understanding of high-lift flows with particular emphasis on the flow affected by the slat edge.

CFD has been used to analyze high-lift systems in many instances, but the vast majority of this work has been confined to two dimensions. Such work has contributed greatly to the understanding of the nature of high-lift flows, and while further two dimensional studies still have much to offer, the study of three-dimensional realistic high-lift flows is of great importance to aircraft designers. A thorough three-dimensional study should investigate how two-dimensional results can be related to the three-dimensional flow phenomena. This is a relatively young field of study, and the development and assessment of the software tools and techniques required to model 3-D flows is vital to the growth of the field. A 3-D computational analysis would enhance the investigation of three-dimensional flow physics, especially in conjunction with similar experimental investigations.

The goal of this study was to apply the current computational tools to a basic three-dimensional high-lift system consisting of a main element with leading slat and trailing flap. In this paper, a description of the investigated geometry, flow solver, turbulence model, and boundary conditions will be presented. The grid strategy developed to resolve flow about a 3-D high-lift system in an efficient manner is discussed. Computations of flow over a three-element wing between wind tunnel walls will be compared with the results of a proven 2-D flow solver to verify the computational approach and results. The results for a 3/4-span slatted wing (also with a half-span flap) will also be presented. Conclusions about the computational approach and the flow physics will be made.

GEOMETRY:

The geometries studied were all based on the NACA 63-215 Mod. B airfoil section, equipped with 30% chord Fowler flap and convex-shaped slat. Figure (1) shows the airfoil section with both slat and flap elements deployed.

The full-span slat case, shown in Figures (5-7), is a simple extrusion of the slat across the leading edge of the wing, so all the spanwise slat stations are identical. The wing fully spans a modeled wind tunnel test section, which is modeled after the NASA Ames 7x10 Foot Wind Tunnel. This full-span slat was added to the three-dimensional grid presented in References (3-5), which describe the full-span wing with half-span flap geometry. The three-quarter span slat case consists of a flapped airfoil over half of the tunnel span, and an unflapped section comprising the second half of the span, with the slat extending 3/4 of the tunnel span from the same wall as the flap extends. This geometry is shown in Figure (8) .

NUMERICAL INVESTIGATION:

FLOW SOLVER:

THE INS3D-UP (Incompressible Navier-Stokes Upwind) flow solver was used for this work (Ref. 6). INS3D-UP is a robust and efficient code that has shown promise in its ability to model three-dimensional high-lift flows in the past (Ref. 1). The two-dimensional version of the code, INS2D-UP (Ref. 7) has demonstrated the ability to accurately predict two-dimensional high-lift flows (Refs. 1, 2). For the current three-element geometry in a landing configuration (i.e. low speed), the exclusion of compressible effects was felt to be a minor sacrifice for the efficiency gained by using an incompressible code. However, if time permits, compressible calculations will be performed and compared to incompressible results, noting any significant changes in predicted flow patterns. INS3D utilizes the method of artificial compressibility to directly couple the flow velocities and pressures by adding a pressure fluctuation term to the continuity equation. This scheme also converts the incompressible Navier-Stokes equations into a hyperbolic form which can be best utilized through the use of an upwind differencing scheme. In this case, the convective flux terms are differenced with a flux-difference splitting scheme based on the method of Roe. The viscous fluxes are discretized using a standard central-difference formulation. The system of equations is solved with a Gauss-Seidel type implicit line relaxation scheme. Convergence for the cases in this study will require approximately on the order of 400 iterations for a grid with 2.25 million grid points. Typical run times are expected to be about 15 Cray C90 ırs.

TURBULENCE MODEL:

The Baldwin-Barth turbulence model was used for all of the cases discussed in this report. It has been shown by a number of researchers that this model does a reasonable job in predicting multi-element airfoil flows, and it was this recommendation, along with its low computational requirements, that strongly suggested the use of this model for this study. The current implementation of turbulence models in INS3D-UP requires that the code be run in either a fully laminar or turbulent mode, therefore, all of the cases described herein will be run in a fully turbulent mode, with no modeling of transition.

COMPUTATIONAL GRIDS:

Structured, overset grids were used throughout this study due to the versatility of the Chimera scheme. Individual grid zones were generated independently and later combined into one multi-grid using the Chimera based PEGSUS (Ref. 8) code. PEGSUS merges the grids into a single file and establishes interpolation links between the grids. This code enables the user to create holes in overlapping meshes and remove (blank) points which would fall outside of the computational domain (such as inside a solid surface).

e slat grid was generated as a 2-D plane, using a C-grid topology, and added grids previously generated by other researchers. Grid details can be seen in Figures (5,6) and are fully described in References (3-5). The wall spacing is approximately 10E-5 chords, while the circumferential spacing is 10E-3 chords at the trailing edge of each element. The size of the 2-D slat grid was 121x60, which was expanded into a 3-D surface by copying spanwise stacks to fill in the third dimension. For the full-span slat case, 45 identical spanwise grid planes were used. Seven zones were required for this simulation: flapped main element, flap, flap-edge, unflapped main element, patch for exposed surface between main element sections, slat, and wind tunnel. The span covers a two chord distance. [illegible]nwise grid density for the slat should have negligible effects on the flow solution, so no further full-span refinements will be attempted. The final full -span slat case grid contained 2.25 million grid points.

Another zone will be added for the part-span slat calculation: the slat-edge grid, which will increase the total grid points to approximately 2.5 million grid points. It is likely that the existing main element grids, which currently contain grid planes clustered about the tunnel mid-span, will need to be modified to include more points in the slat-wake region in order to clarify flow characteristics, which will further increase total grid points and calculation time.

Small scale flow features such as secondary separation lines are expected to be visible in the solutions, with adequate grid resolution. Variations of grid resolution will be performed in order to develop an adequately resolved 3/4-span slat solution.

BOUNDARY CONDITIONS:

Most solid surface points were modeled with a no-slip boundary condition, which specifies zero velocity and a zero normal pressure gradient at the surface. Other solid surfaces were specified as Chimera boundary points and were allowed to be updated through interpolation (Ref. 4). Wake-cut boundary conditions were needed for elements with C-grid topology, and those points were updated by first order averaging of points on either side of the wake cut. The wind tunnel walls were represented by slip-walls which impose a zero normal gradient for all flow variables. The tunnel inflow condition was prescribed with free stream flow, while the tunnel outflow consisted of constant static pressure and extrapolated velocity boundary conditions. The outer boundaries of all grids inside the wind tunnel grid were Chimera boundaries.

RESULTS AND DISCUSSION:

The following section highlights some of the areas to be discussed in the paper. All of the accompanying figures are draft copies only.

TWO-DIMENSIONAL MULTI-ELEMENT AIRFOIL:

A two-dimensional grid study concerning the effects of slat location was performed. Figure (1) shows the layout and overset of the three element grids. The outer boundary of the main element grid extends approximately 20 chord lengths in all directions, and solutions were found for free stream flow conditions. Figure (2) shows the pressure distribution about the airfoil, contours colored by pressure. The pressure distribution is plotted in Figure (3) for all three elements. The slat gap (the vertical distance between increased in increments of 0.002 inches, and the resulting pressure [illegible]stributions are shown in Figure (4), where it can be seen that changing the [illegible]p size resulted in nominal differences in pressure over the elements. The numerical values of the lift coefficient did not vary significantly either, thus it was decided to leave the slat in its original position. These solutions were calculated at a Reynolds number of 3.7 million and an angle of attack of 10 degrees.

FULL-SPAN SLAT:

' full-span results will be presented as a validation case. The computational results are for a free stream Reynolds number of 3.7 million and an angle of attack of 10 degrees. The computational pressures and forces will be analyzed, and small three-dimensional effects seen in the computations will be discussed. The flows computed with INS2D-UP and INS3D-UP will be compared. A cross-section of the unflapped main element with slat is shown in Figure (5), a cross-section of the three-element slat-main-flap combination is shown in Figure (6), and a three-dimensional representation of the multi-element airfoil is given in Figure (7).

3/4-SPAN SLAT:

The majority of the results in the proposed paper will be focused on the three-quarter span slat computations, which are also performed for a Reynolds number of 3.7 million and an angle of attack of 10 degrees. A grid study will be performed to determine how best to resolve slat-wake flow over the main element and half-span flap. The lift distribution will be computed, and compared to the full-span slat case results to determine the effect that the finite span slat has on the total lift. The lift distribution results for several cross-sections of this case will be compared to the two-dimensional results to better illustrate the complexity of the flow field. A representation of the 3/4-span slat, main element, half-span flap combination is given in Figure (8).

In addition, several types of flow visualization will be implemented to obtain ualitative understanding of the flow features (such as streamline curvature, separation lines, tip vortex formation, and wake position). The nature of the flow will be clearly seen through the use of flow visualization.

Experimental analyses of slat-edge and slat-wake effects on an airfoil with a half-span flap are scheduled in the NASA Ames 7x10 Wind Tunnel for spring 1996 for configurations similar to those generated in this study.

REFERENCES:

1. Rogers, S., "Efficient Simulation of Incompressible Viscous Flow Over Single and Multi-Element Airfoils", AIAA Paper 92-0405, Jan. 1992.

2. Rogers, S., "Progress in High-Lift Aerodynamic Calculations", AIAA Paper 93-0194, Jan. 1993.

3. Mathias, D., "Navier-Stokes Analysis of the Flow About a Flap Edge", Master of Science Thesis, California Polytechnic State University, San Luis Obispo, CA, 1994.

4. Mathias, D., Roth, K., Ross, J., Rogers, S., and Cummings, R., "Navier-Stokes Analysis of the Flow About a Flap Edge", AIAA Paper 95-0185, AIAA 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 1995.

5. Mathias, D., Roth, K., Ross, J., Rogers, S., and Cummings, R., "Computational Investigation of a Semi-Span Flap", 6th International Symposium on Computational Fluid Dynamics, Lake Tahoe, NV, Sept. 1995.

6. Rogers, S., "Numerical Solution of the Incompressible Navier-Stokes Equations", NASA TM 102199, Nov. 1990.

7. Rogers, S., and Kwak, D., "Upwind Differencing Scheme for the Time-Accurate Incompressible Navier-Stokes Equations", AIAA Journal, Vol. 28, No. 2, Feb. 1990.

8. Tramel, T, and Suhs, J., "PEGSUS 4.0 User's Manual", AEDC TR-91-8, June 1991.

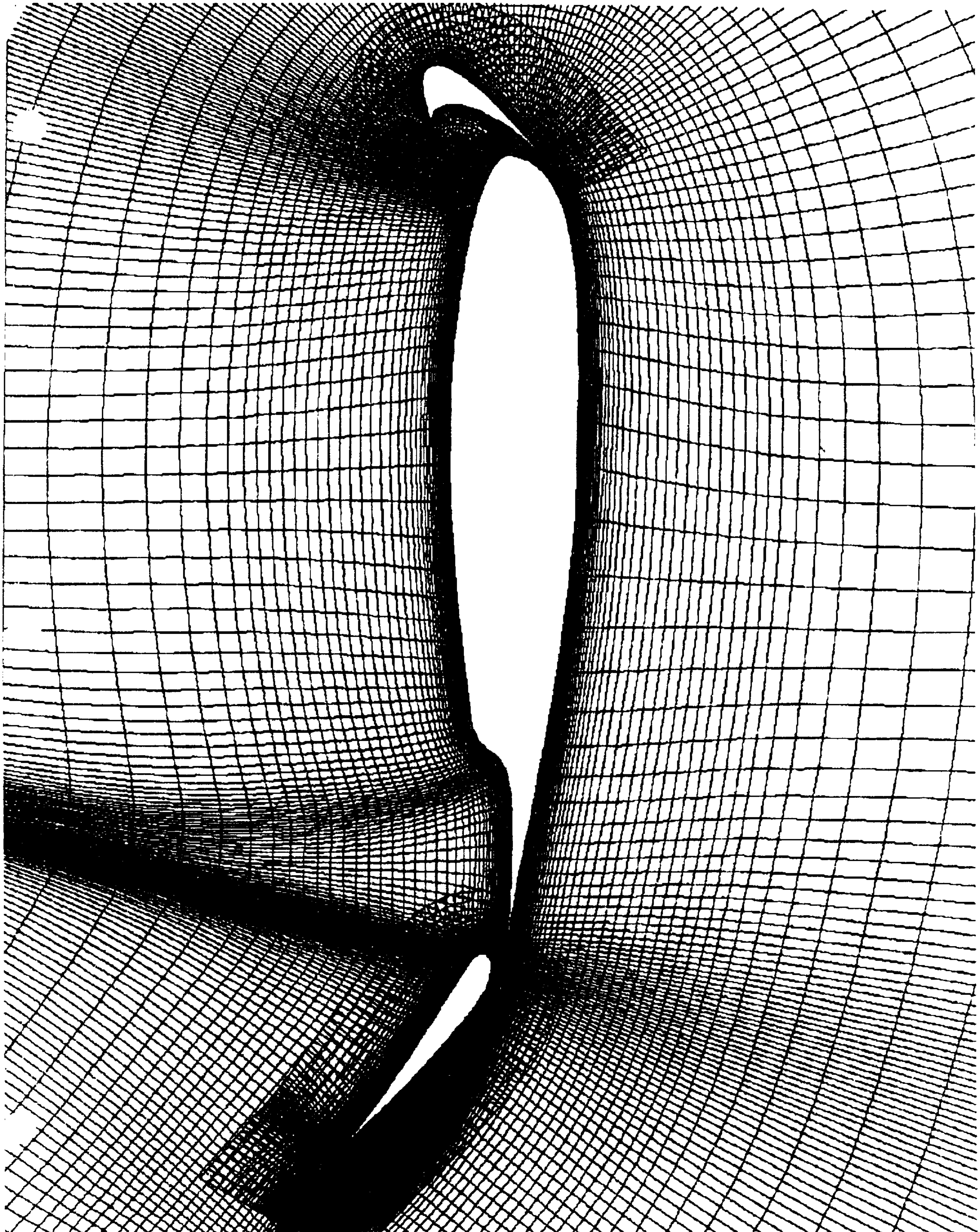

Figure 1: Two-dimensional multi-element grid

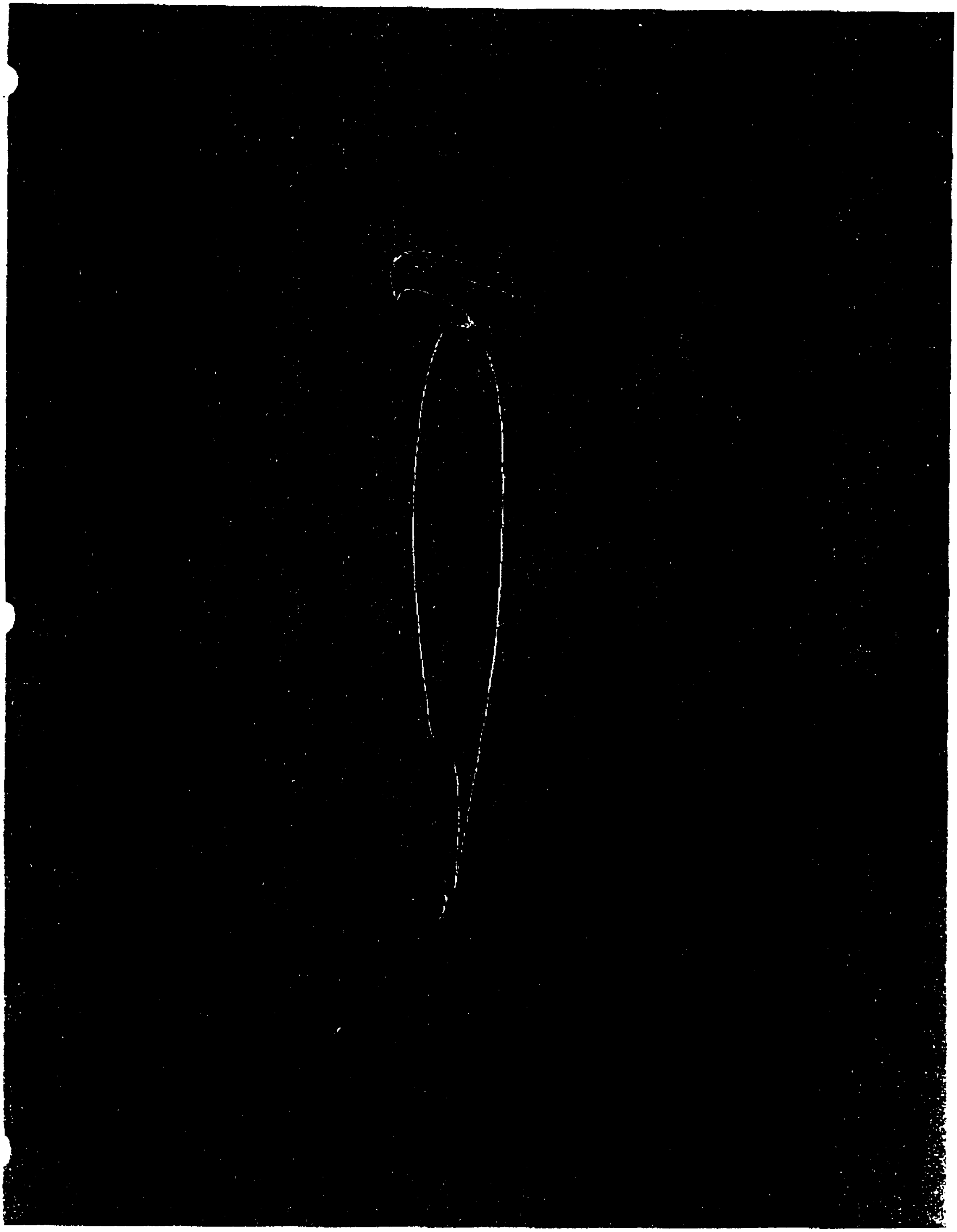

Figure 2: Pressure Contours over Two-Dimensional Multi-element grid

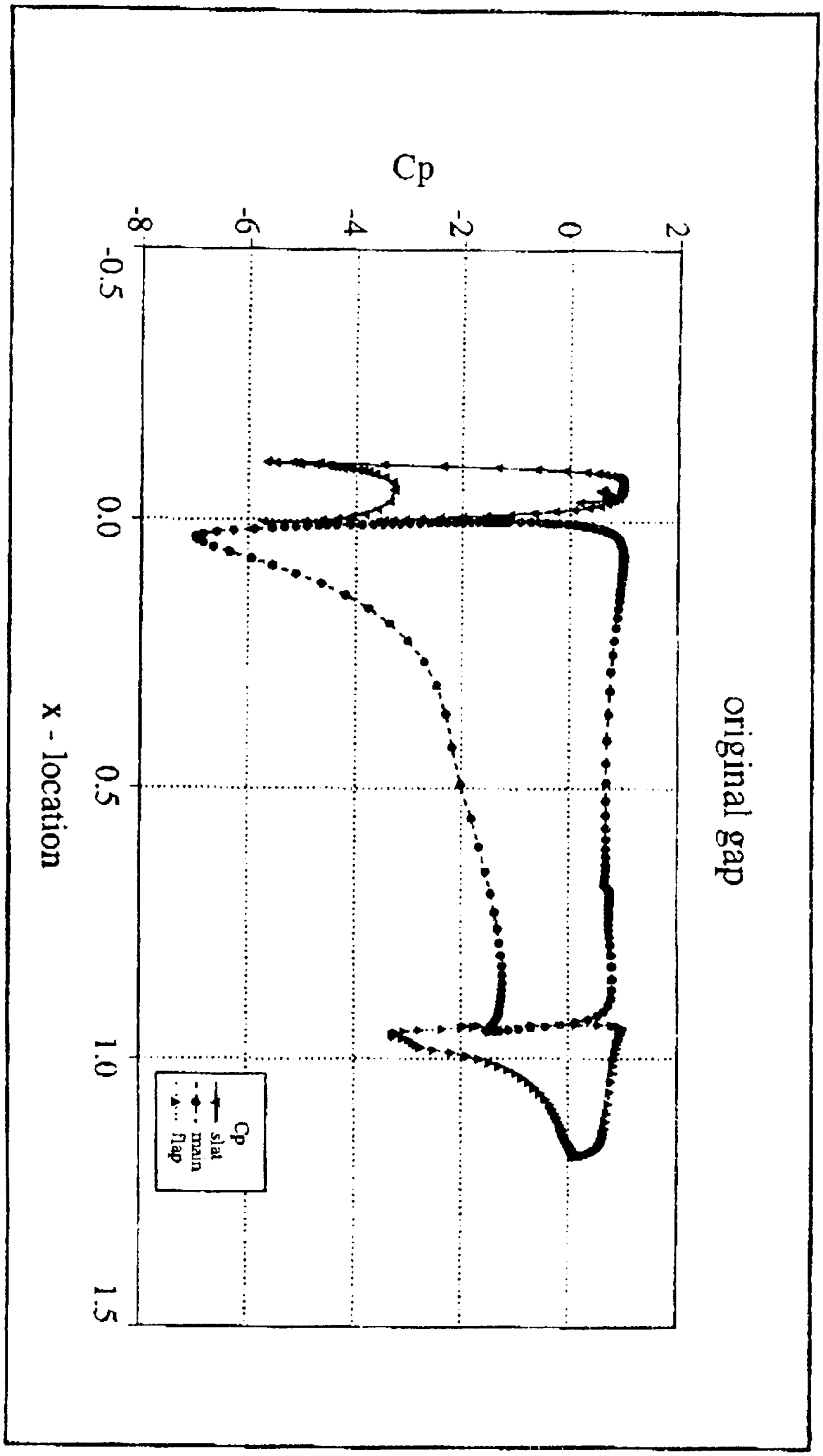

Figure 3: Pressure Distribution over Two-Dimensional Multi-element Airfoil

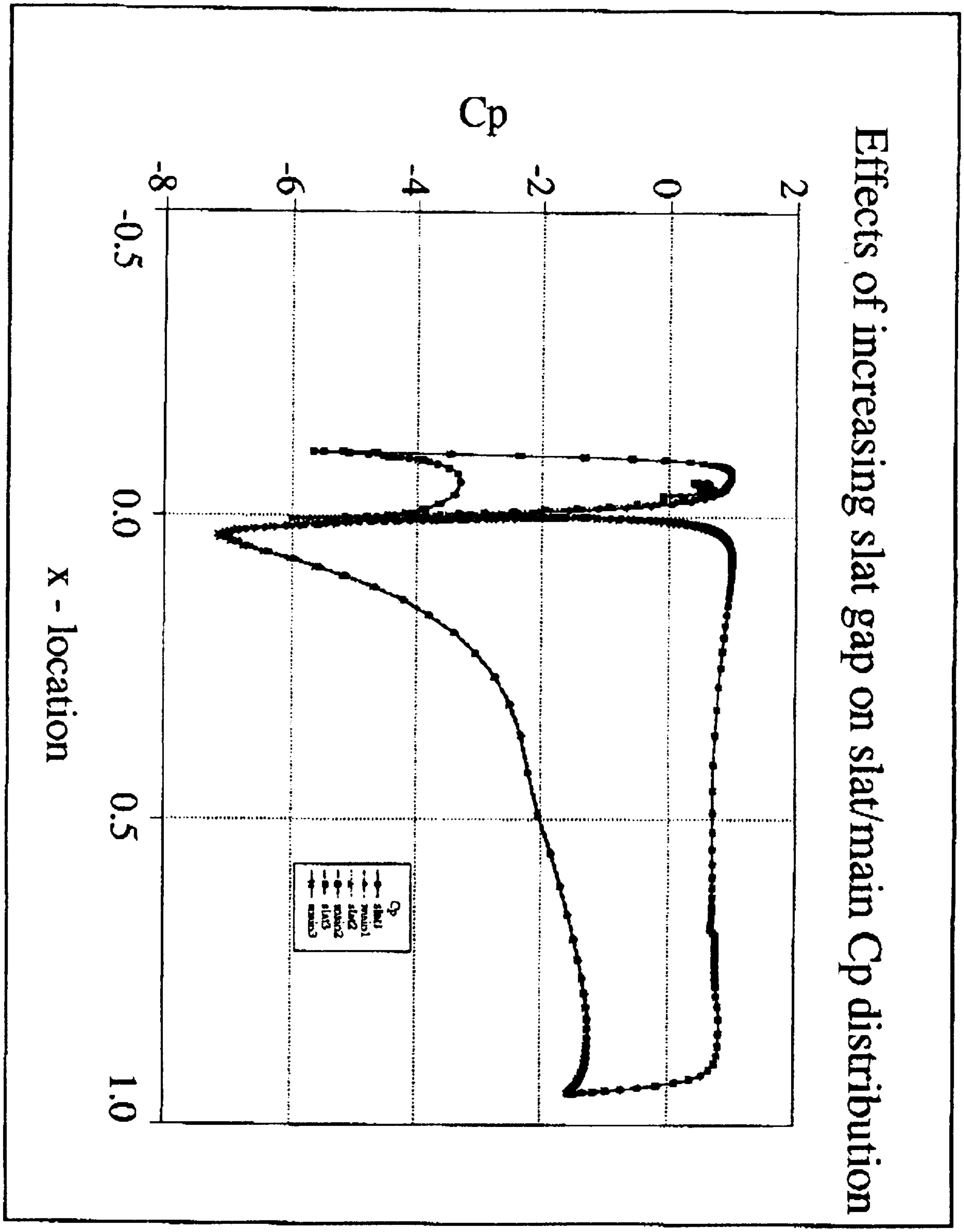

Figure 4: Comparison of Pressure Distributions for variation of Slat Gap (2-D)

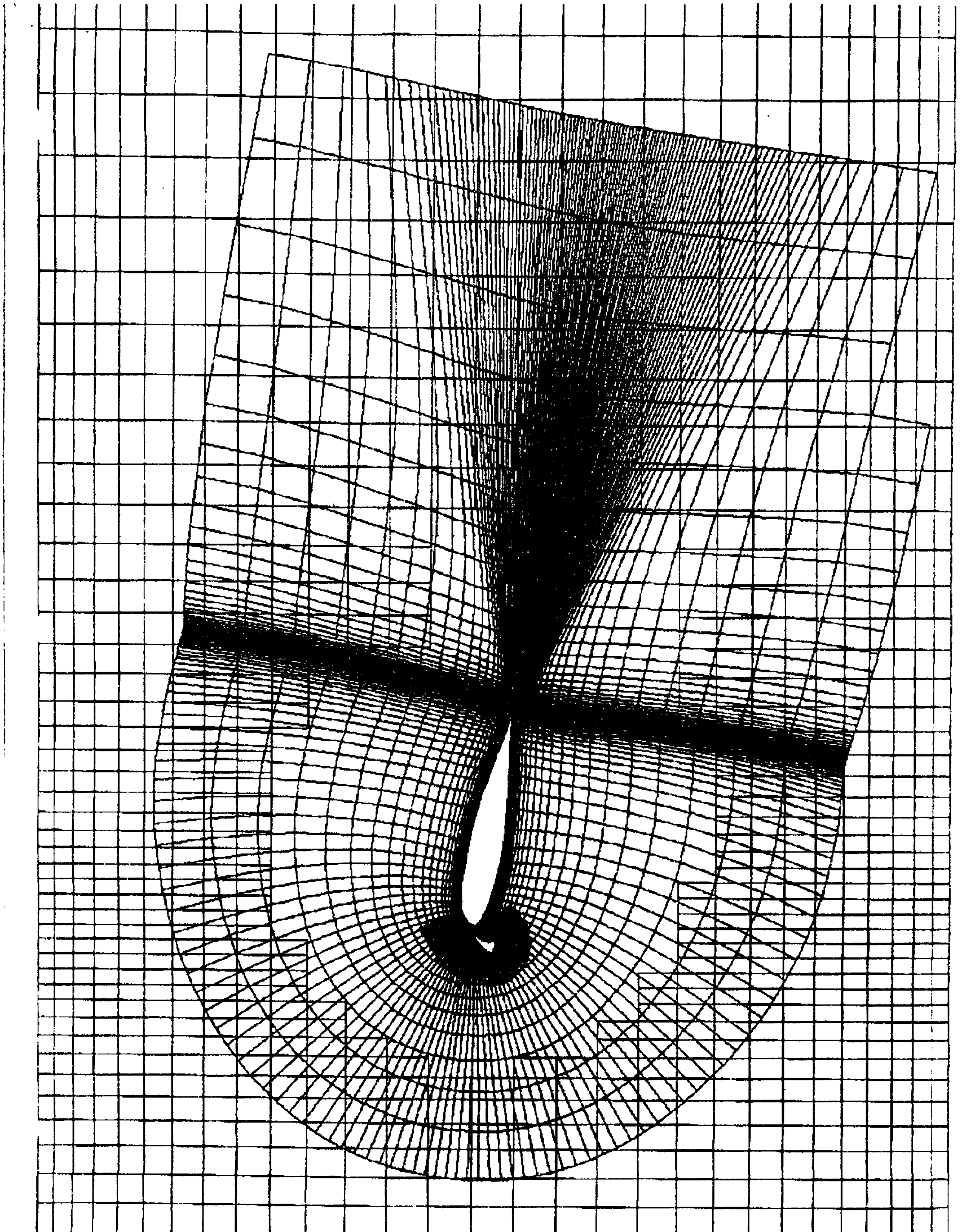

Figure 5: Unflapped Airfoil Grid Cross Section

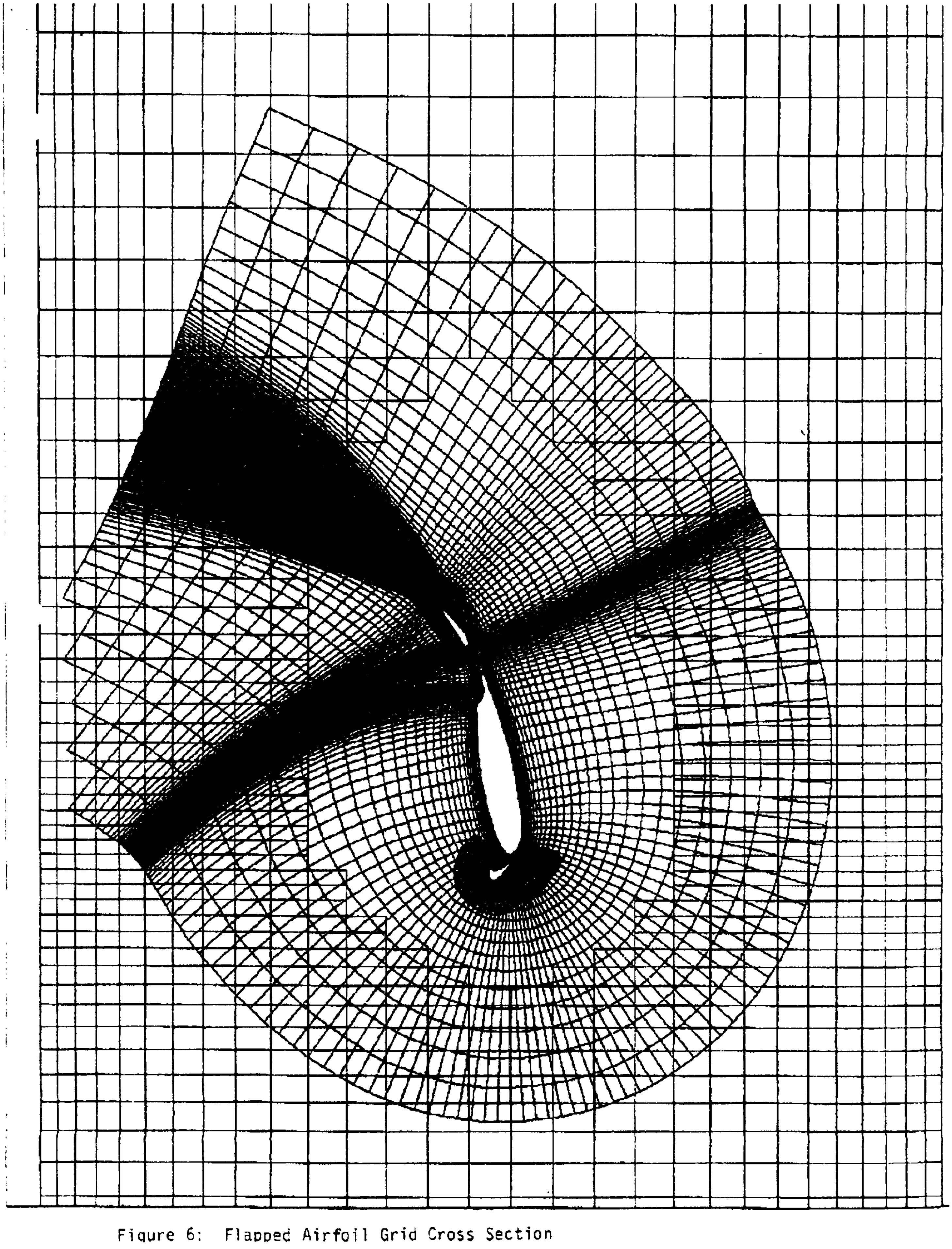

Figure 6: Flapped Airfoil Grid Cross Section

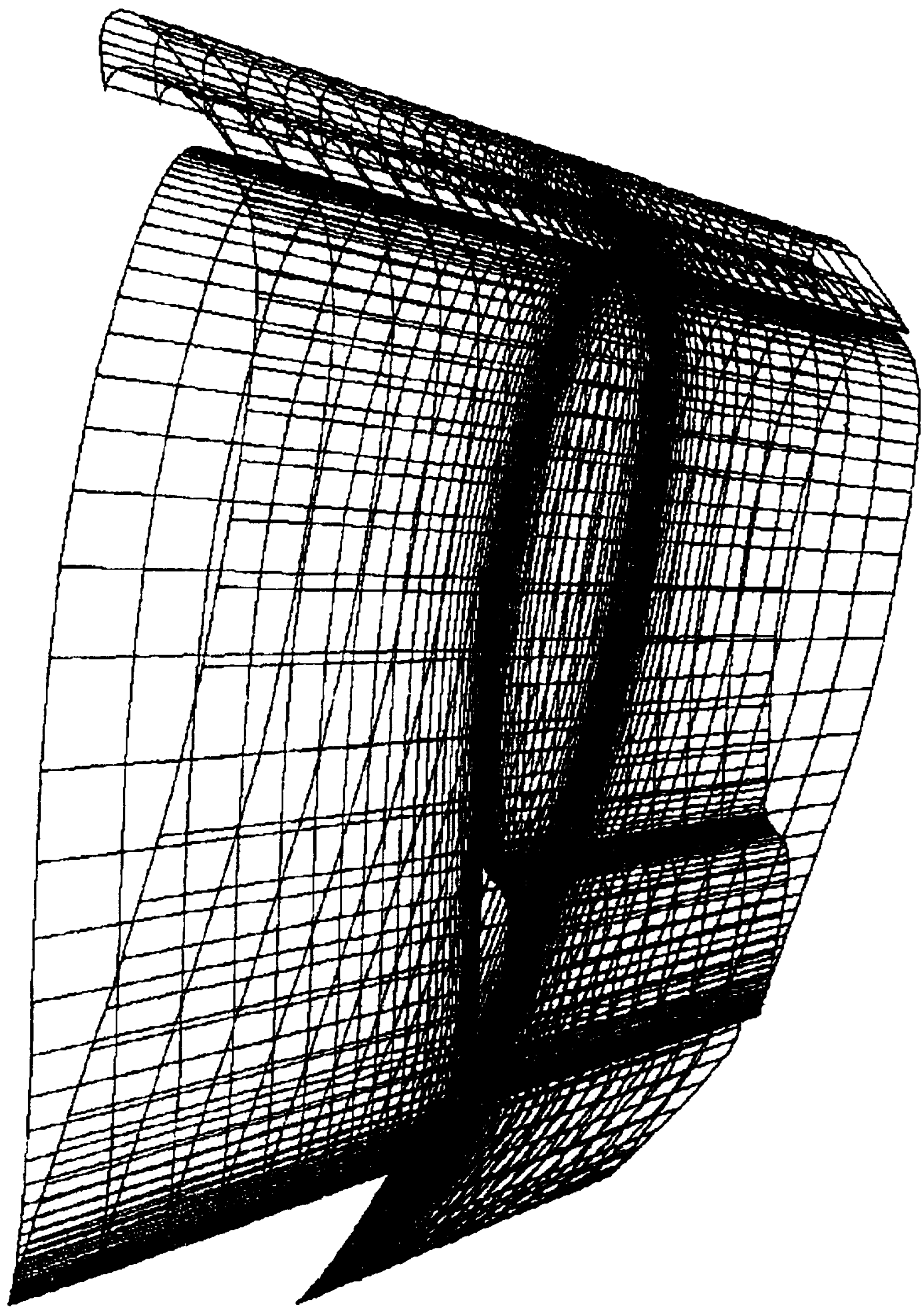

Figure 7: 3-D, Full-span Slat Grid Surface Representation

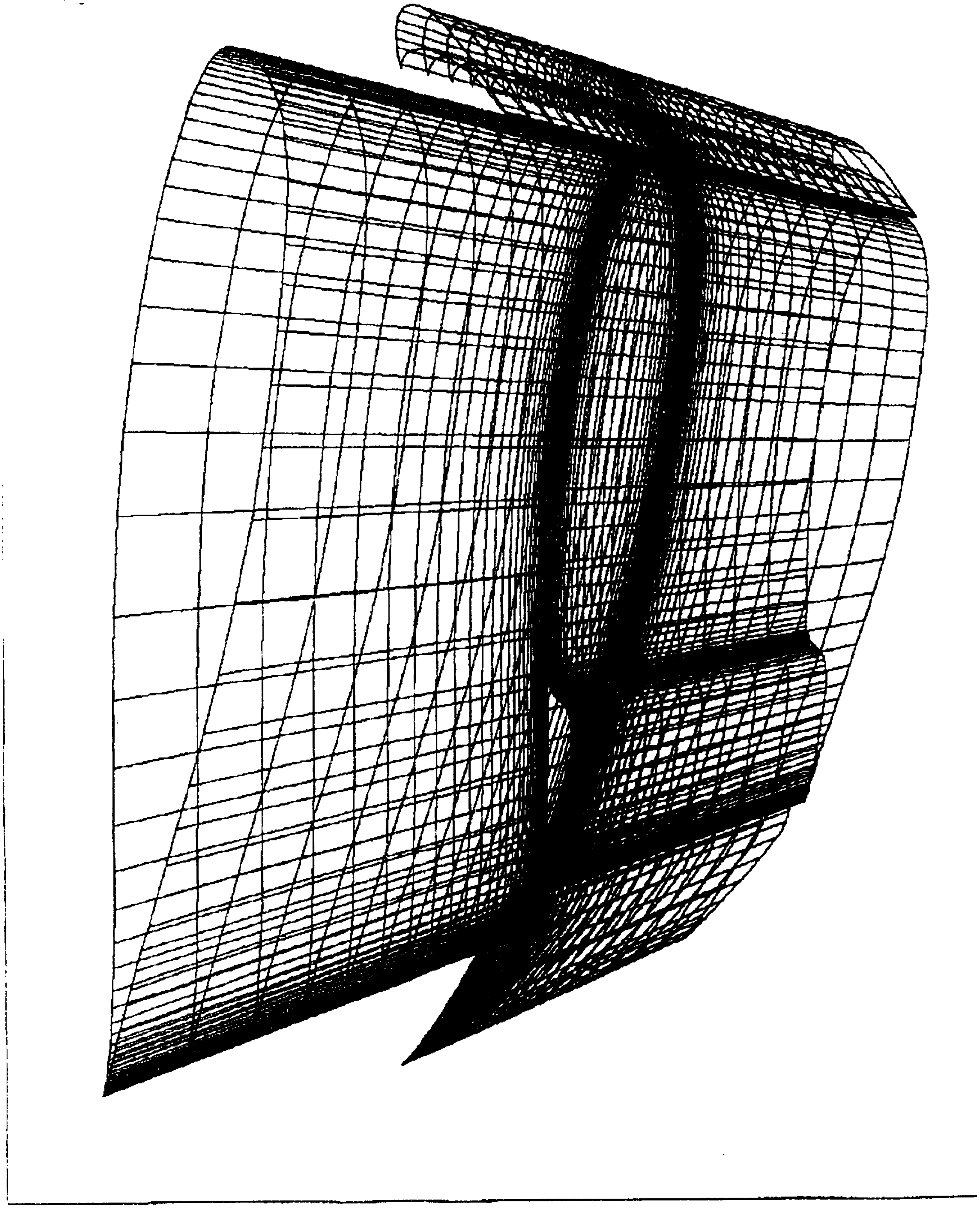

Figure 9. 3-D, 3/4 span Slat Grid Surface Representation