

NASA Technical Memorandum 106371  
AIAA-94-0391

1N-02  
191163  
9P

# Flow Analysis for the Nacelle of an Advanced Ducted Propeller at High Angle-of-Attack and at Cruise with Boundary Layer Control

D.P. Hwang, D.R. Boldman, and C.E. Hughes  
*Lewis Research Center*  
*Cleveland, Ohio*

Prepared for the  
32nd Aerospace Sciences Meeting and Exhibit  
sponsored by the American Institute of Aeronautics and Astronautics  
Reno, Nevada, January 10-13, 1994

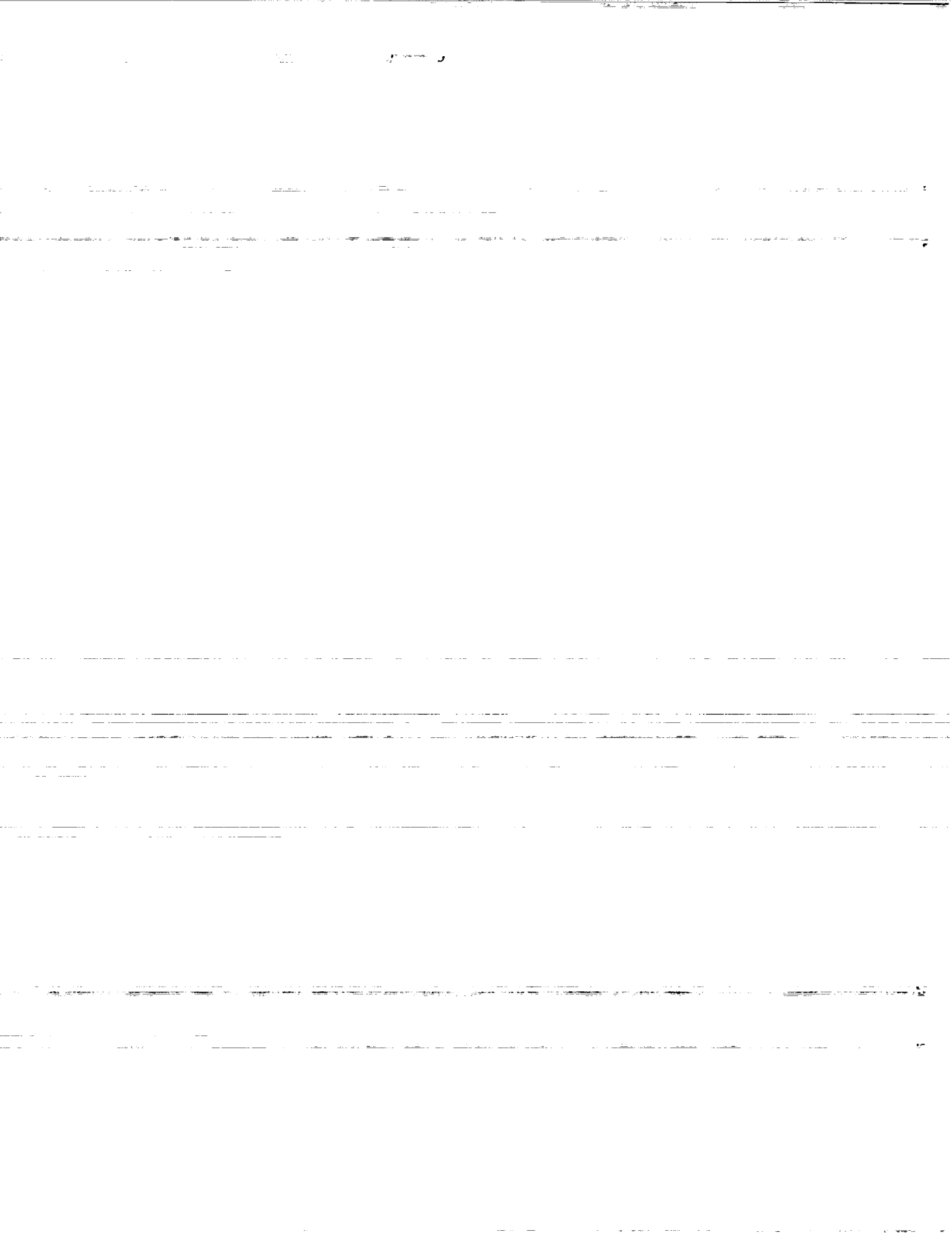
(NASA-TM-106371) FLOW ANALYSIS FOR  
THE NACELLE OF AN ADVANCED DUCTED  
PROPELLER AT HIGH ANGLE-OF-ATTACK  
AND AT CRUISE WITH BOUNDARY LAYER  
CONTROL (NASA) 9 p

N94-16571

Unclass

63/02 0191163

**NASA**



Flow Analysis for the Nacelle of an Advanced Ducted Propeller at High  
Angle-of-Attack and at Cruise with Boundary Layer Control

Danny P. Hwang,\* Donald R. Boldman,† and Christopher E. Hughes\*  
National Aeronautics and Space Administration  
Lewis Research Center  
Cleveland, Ohio 44135

Abstract

An axisymmetric panel code and a three-dimensional Navier-Stokes code (used as an inviscid Euler code) were verified for low speed, high angle-of-attack flow conditions. A three-dimensional Navier-Stokes code (used as an inviscid code), and an axisymmetric Navier-Stokes code (used as both viscous and inviscid code) were also assessed for high Mach number cruise conditions. The boundary layer calculations were made by using the results from the panel code or Euler calculation. The panel method can predict the internal surface pressure distributions very well if no shock exists. However, only Euler and Navier-Stokes calculations can provide a good prediction of the surface static pressure distribution including the pressure rise across the shock. Because of the high CPU time required for a three-dimensional Navier-Stokes calculation, only the axisymmetric Navier-Stokes calculation was considered at cruise conditions. The use of suction and tangential blowing boundary layer control to eliminate the flow separation on the internal surface was demonstrated for low free stream Mach number and high angle-of-attack cases. The calculation also shows that transition from laminar flow to turbulent flow on the external cowl surface can be delayed by using suction boundary layer control at cruise flow conditions. The results were compared with experimental data where possible.

Symbols

$C_f$	skin friction
$C_p$	pressure coefficient
$c$	chord length
$\dot{w}$	corrected weight flow
$x$	$x$ coordinate
$\alpha$	angle-of-attack

Introduction

The progress in computational fluid dynamics in recent years makes it possible to use CFD codes as part of the design process. So far the panel method with a boundary layer calculation<sup>1-4</sup> has been used as a design tool for subsonic inlets. It is an efficient and economical way to obtain a good solution for internal subsonic flow without separated flow and shock formation; however, this method is limited to the internal flow of the inlets.<sup>5-8</sup> A three-dimensional Navier-Stokes solution, on the other hand, can also be obtained for a subsonic inlet;<sup>9</sup> however, the CPU time of 50 hr or more for one calculation on a Cray-YMP make it impractical for use as a design tool. For a high angle-of-attack case, a three-dimensional Euler/boundary layer approach can be used. The CPU time on a Cray-YMP for this approach is about 25 min which is reasonable for design applications. An axisymmetric Navier-Stokes code can also be used for an ADP nacelle at cruise with less than 15 min of CPU time on a Cray-YMP. Because of the effect of downstream conditions on the upstream flow field in subsonic flow, the best results are obtained by combining the inlet and the nozzle as an integrated geometry and performing the computation on both at the same time.

In this paper, the axisymmetric panel/boundary layer method and a three-dimensional Euler/boundary layer approach were applied to a low speed, high angle-of-attack case, and the axisymmetric Euler/boundary layer approach and an axisymmetric Navier-Stokes calculation were applied to an ADP nacelle at cruise. The computational results are compared with experimental data.

Experiments

An ADP inlet/nacelle has been tested extensively with a 17-in. powered fan simulator in the NASA Lewis 9- by 15-Foot Low Speed Wind Tunnel (LSWT) and 8- by 6-Foot Supersonic Wind Tunnel (SWT) as part of a joint effort between NASA Lewis Research Center and Pratt & Whitney (P&W).<sup>10,11</sup> The experimental data

\*Senior Research Engineer, Member, AIAA.

†Senior Research Engineer, Associate Fellow, AIAA.

provided an excellent database for computational analysis verification. The test configuration which was called the conventional inlet was selected for the comparison of analytical results with experimental data. A sketch of this configuration is shown in Fig. 1. The conventional inlet was the longest of a series of inlets designed by P&W and tested in the NASA wind tunnels.

### Numerical Methods

The panel code starts with a geometry program which creates the control points for computation. The incompressible potential flow program is then used to calculate the basic solutions to the problem.<sup>1-3</sup> These solutions are used to provide a solution that satisfies the inlet operating conditions of freestream velocity, angle-of-attack, and inlet mass flow. Next, the incompressible flow is corrected for compressibility effects.<sup>1</sup> The compressible potential flow solution is then used as an input to the boundary layer program<sup>4</sup> which calculates the laminar, transition and turbulent boundary layer characteristics which can include the effects of suction and tangential blowing boundary layer control. There are three versions of the panel code; namely, two-dimensional, axisymmetric, and three-dimensional.

The Navier-Stokes codes, PARC3D and PARC2D/AXI,<sup>12,13</sup> were used to obtain the Euler solutions by turning off all viscosity terms (in the Euler mode). Both codes use the Beam and Warming approximate factorization algorithm. This algorithm is an implicit scheme which solves the set of equations produced by central-differencing the Navier-Stokes equations on a regular grid. Since these equations are formulated in the strong conservation form for a curvilinear set of coordinates, the algorithm is quite general. The artificial dissipation terms were added for improving stability and reducing the oscillations in the solution due to central differencing. It includes a multi-blocked scheme so that it can handle complicated geometries. In addition, boundary conditions can be easily specified by the user such that it can be called a user-friendly code.

The surface flow information of the Euler solutions were used as an input to the TRACEON code<sup>3</sup> which traces a three-dimensional streamline on the nacelle surface. The flow information on this streamline is then used as an input to the boundary layer program<sup>4</sup> to obtain the flow characteristics such as skin friction, laminar/turbulent transition, etc.

### Computational Grids

The two- and three-dimensional computational grids were generated with GRIDGEN2D.<sup>14</sup> The two-

dimensional grid was rotated with respect to the x-axis to create an axisymmetric three-dimensional grid. Although this report is for an axisymmetric conventional ADP nacelle, the three-dimensional codes are available for a three-dimensional nacelle.

The computational grids for the Euler calculation, are shown in Fig. 2. Figure 2(a) shows the two-dimensional grid with four blocks for the axisymmetric calculation. Only 10 200 grid points were needed for the axisymmetric Euler computation. For the axisymmetric Navier-Stokes calculation, only the C-grid around the nacelle was regridded for the viscous computation, and the other three blocks were kept the same as the Euler grid and computed in the Euler mode. By this zonal method, the total grid points, even for the Navier-Stokes calculation, was kept at 13 030. Figure 2(b) is the grid with five blocks for the high angle-of-attack computation. A C-grid was wrapped around the nacelle for better resolution in the area of high curvature near the highlight (the leading edge of a nacelle). An H-grid was used for the other part of the computation. A total of 120 000 grid points were needed for the three-dimensional Euler grid. In order to obtain a good prediction of external static pressures from the leading edge up to the trailing edge of the nacelle, two blocks were used for both grids to compute the exhaust plume from the nozzle.

### Results

#### Low Mach number, high angle-of-attack operation

Figure 3 shows the pressure coefficients on the surface for a free stream Mach number of 0.2, an angle-of-attack of 25 deg and  $\dot{w} = 17.4$  kg/sec. Although the pressure peaks in the computational results were slightly different from experimental data, the overall prediction for the high angle-of-attack using both panel code and PARC3D code in Euler mode was fairly good.

Figure 4 shows the prediction of separated flow on the internal surface at a Mach number of 0.2 and an angle-of-attack of 25 deg by using the panel/boundary layer method and the three-dimensional Euler/boundary layer approach. The separation point was predicted at the point where the skin friction coefficient,  $C_f$ , was equal to zero. Both methods predicted the identical separation point at  $x/c = 0.264$ .

By using either suction boundary layer control or tangential blowing boundary layer control, the flow separation on the internal surface could be eliminated for the high angle-of-attack case as shown in Figs. 5(a) and (b). Herring's boundary layer code<sup>4</sup> was used for the computation. A very small region of suction, applied from  $x/c = 0.258$  to  $x/c = 0.269$  with a bleed rate of

0.03 percent of the captured mass flow rate, could remove flow separation entirely as shown in Fig. 5(a). The internal flow separation also could be eliminated by tangential blowing at  $x/c = 0.242$  with a slot height of 0.5 mm and a blowing mass flow rate of 0.12 percent of the captured mass flow rate as shown in Fig. 5(b).

### Cruise performance

The pressure coefficients on the internal and external surfaces of the nacelle are shown in Fig. 6 for a cruise Mach number of 0.85. The results include the axisymmetric Euler (PARC2D/AXI) calculation and three-dimensional Euler (PARC3D) calculation. Excellent agreement between the Euler solutions and experimental data was obtained except near the shock location. The shock location is evident in the pressure distribution at an  $x/c$  of about 0.264. In this calculation, the PARC2D/AXI code captured the shock better than the PARC3D code.

Figure 7 shows the comparison of the pressure coefficients between the Euler solution and the Navier-Stokes results at the cruise condition. The analytical result from the viscous Navier-Stokes calculation compared more favorably than the Euler calculation. The shock was captured by both Euler and Navier-Stokes calculations. In addition, the Navier-Stokes result eliminated the overshoot predicted by the Euler calculation at about  $x/c = 0.3$ .

The transition from laminar flow to turbulent flow at cruise conditions can be predicted by finding the critical point where the displacement thickness Reynolds number of the boundary layer first becomes greater than the critical Reynolds number. Herring's boundary layer program<sup>4</sup> was used. Based on this method, the transition point for the ADP nacelle at cruise conditions is located at about  $x/c = 0.08$ , where  $x$  is the distance from the highlight and  $c$  is the length of the nacelle. The transition point could be moved further downstream to  $x/c = 0.35$  as shown in Fig. 8 by using suction boundary layer control, applied from  $x/c = 0.006$  to  $x/c = 0.097$  on the external surface. For this case, the suction mass flow rate was 0.7 percent of the captured mass flow rate.

### Concluding Remarks

An axisymmetric panel code and a three-dimensional Euler code were verified for low speed, high angle-of-attack flow conditions by comparing with experimental data. The numerical results from an axisymmetric Navier-Stokes code in both viscous and inviscid modes compared favorably with experimental

data for high Mach number cruise conditions. The boundary layer calculations were made using the results from the panel code or Euler calculation. These calculations demonstrated that both suction and blowing boundary conditions can be applied effectively to prevent internal flow separation in the inlet. The results also suggested that the suction boundary layer control is an efficient way to delay the transition point on the external cowl surface at cruise. These results provide confidence in using these approaches as part of the design process for a nacelle of an ADP propulsion system.

### References

- <sup>1</sup>Stockman, N.O. and Farrell, Jr., C.A., "Improved Computer Programs for Calculating Potential Flow in Propulsion System Inlets," NASA TM-73728, July 1977.
- <sup>2</sup>Hess, J.L. and Friedman, D.M., "Calculation of Compressible Flow in and About Three-Dimensional Inlets With and Without Auxiliary Inlets by a Higher-Order Panel Method," NASA CR-168009 October 1982.
- <sup>3</sup>Hess, J.L., Friedman D.M., and Clark, R.W., "Calculation of Compressible Flow About Three-Dimensional Inlets With Auxiliary Inlets, Slats and Vanes by Means of a Panel Method," NASA CR-174975, June 1985.
- <sup>4</sup>Herring, H.J., "PL2-A Calculation Method for Two-Dimensional Boundary Layers With Cross-flow and Heat Transfer," Dynalysis of Princeton, Report No. 65, 1981.
- <sup>5</sup>Hwang, D.P. and Diedrich, J.H., "A Summary of V/STOL Inlet Analysis Methods," NASA TM-82725, Dec. 1981.
- <sup>6</sup>Hwang, D.P. and Abbott, J.M., "A Summary of V/Stol Inlet Analysis Methods," NASA TM-82885, Aug. 1982.
- <sup>7</sup>Hwang, D.P., "A Numerical Analysis Applied to High Angle-of-Attack Three-Dimensional Inlets," TM-87298, June 1986.
- <sup>8</sup>Boldman, D.R., Iek, C., Hwang, D.P., Jeracki, R.J., Larkin, M., and Sorin, G., "Evaluation of Panel Code Predictions With Experimental Results of Inlet Performance for a 17-Inch Ducted Prop/Fan Simulator Operating at Mach 0.2," NASA TM-104428, June 1991.
- <sup>9</sup>Iek, C., Boldman, D.R., and Ibrahim, M., "Analysis of an Advanced Ducted Propeller Subsonic Inlet," NASA TM-105393, Jan. 1992.
- <sup>10</sup>Larkin, M.J. and Schweiger, P.S., "Ultra High Bypass Nacelle Aerodynamics, Inlet Flow-Through High Angle-of-Attack Distortion Test," NASA CR-189149, July 1992.

- <sup>11</sup>Boldman, D.R., Iek, C., Hwang, D.P., Larkin, M., and Schweiger P., "Effect of a Rotating Propeller on the Separation Angle-of-Attack and Distortion in Ducted Propeller Inlets," NASA TM-105935, Jan. 1993.
- <sup>12</sup>Pulliam, T.H. and Steger, J.L., "Implicit Finite-Difference Simulations of Three-Dimensional Compressible Flow," AIAA Journal, Vol. 18, No. 2, Feb. 1980, pp. 159-167.
- <sup>13</sup>Cooper, G.K. and Sirbaugh, J.R., "PARC Code: Theory and Usage," AEDC-TR-89-15, July 1989.
- <sup>14</sup>Steinbrenner, J.P., Chawner, J.P., and Fouts, C.L., "The GRIDGEN3D Multiple Block Grid Generation System, Volume II: Users' Manual," WRDC-TR-90-3022, Vol. II, July 1990.
- <sup>15</sup>Henderson, T.L., Choo, Y.K., and Lee K.D., "Interactive Solution-Adaptive Grid Generation Procedure," NASA TM-105432, April 1992.

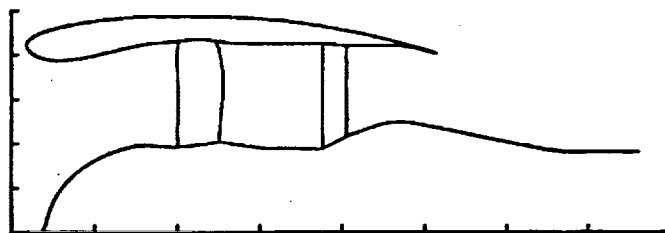
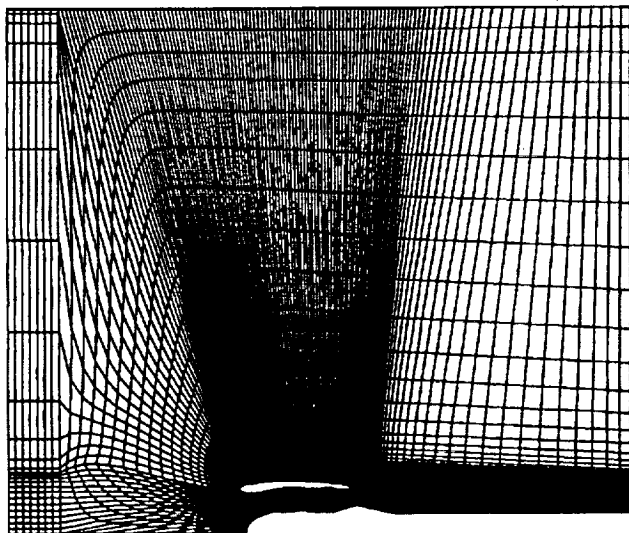
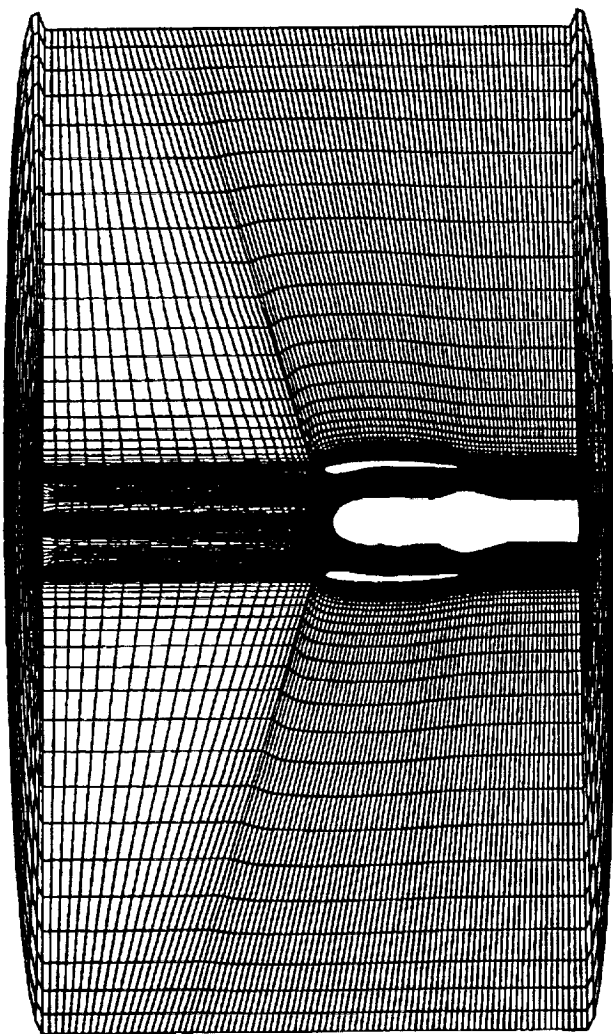


Figure 1.—Geometry of conventional ADP Inlet.



(a) 2-D grid for cruise computation.



(b) 3-D axisymmetric grid for high angle-of-attack.

Figure 2.—Computational grids.

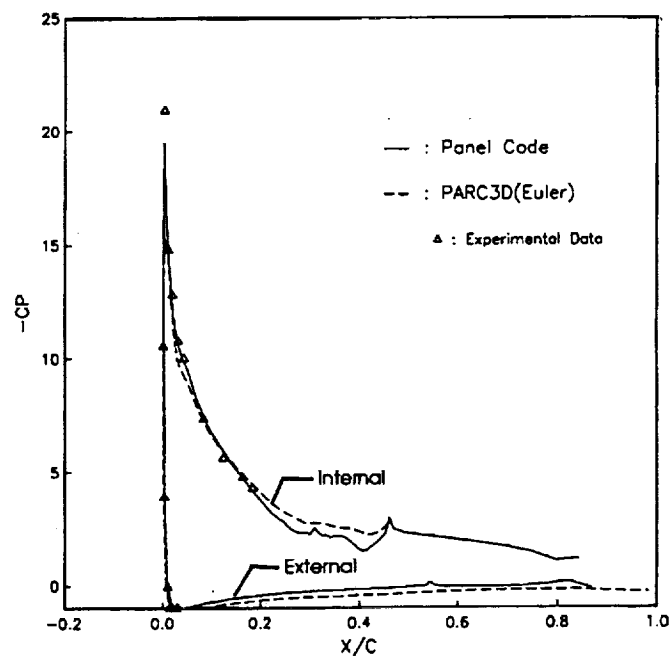


Figure 3.—Pressure coefficient, Mach number = 0.2,  $\alpha = 25^\circ$ ,  $\dot{w} = 17.4$  kg/sec.

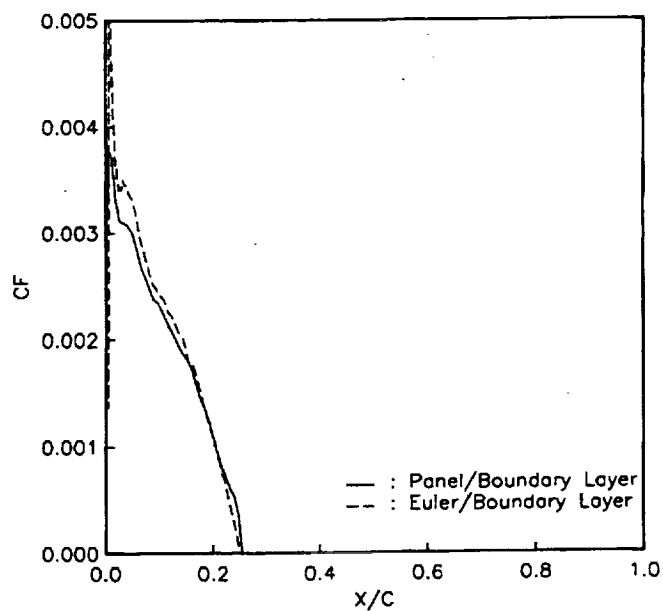
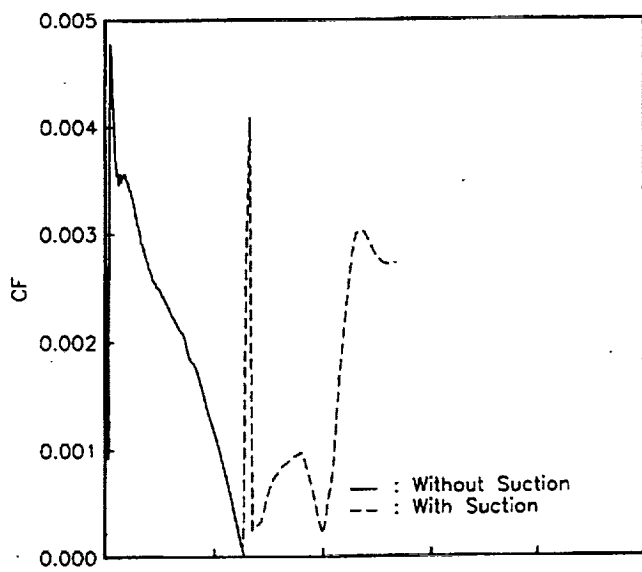
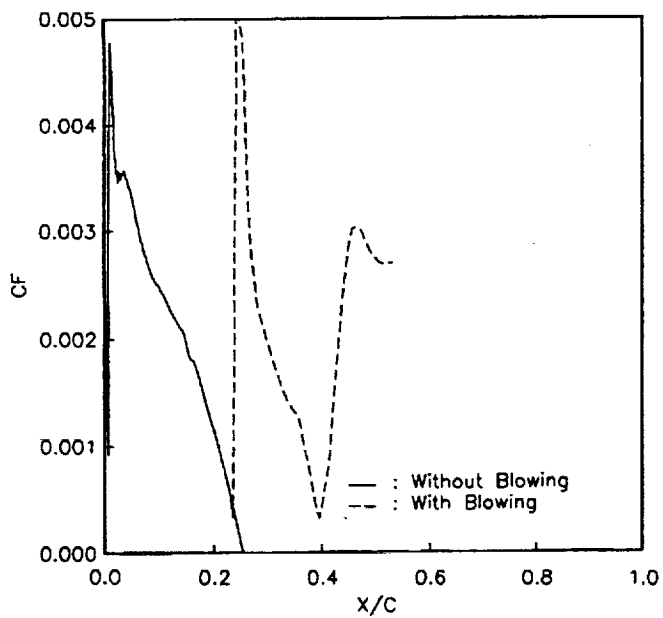


Figure 4.—Prediction of flow separation, Mach number = 0.2,  $\alpha = 25^\circ$ ,  $\dot{w} = 17.4$  kg/sec.



(a) Suction boundary layer control.



(b) Blowing boundary layer control.

Figure 5.—Boundary layer control for a high angle-of-attack case.

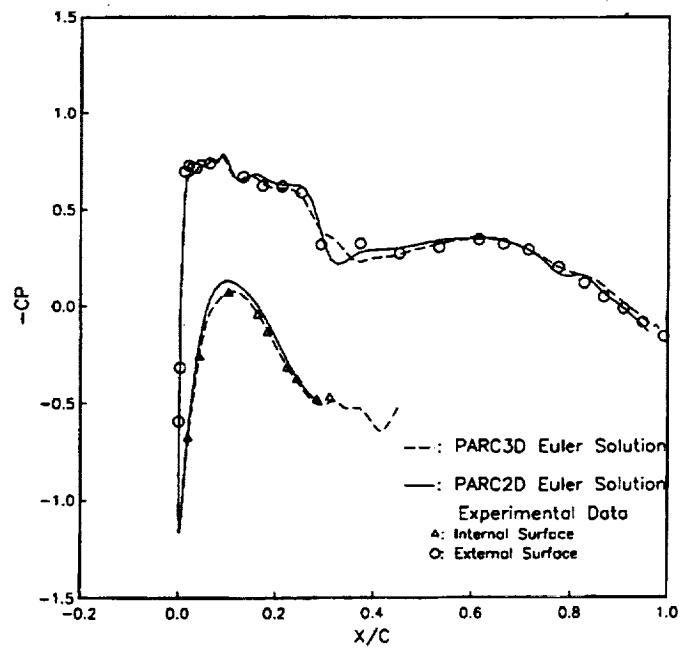


Figure 6.—Pressure coefficient at cruise, Mach number = 0.85.



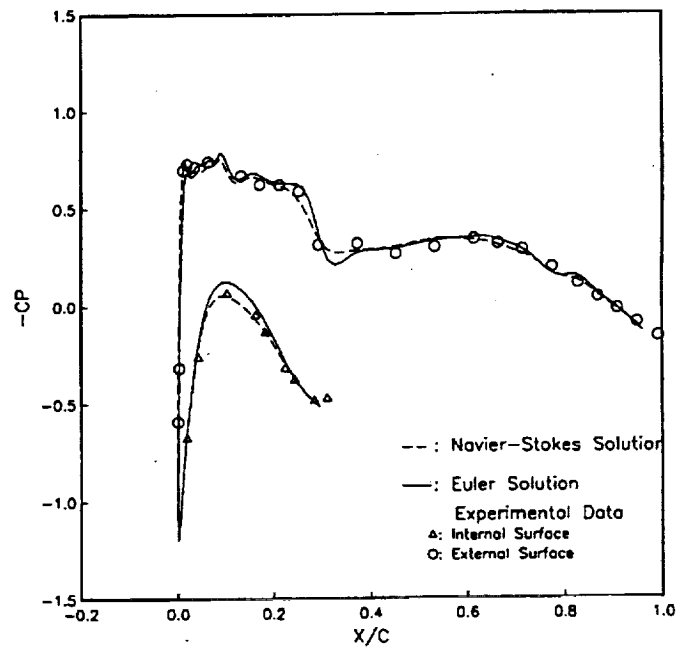


Figure 7.—Euler and Navier-Stokes solutions at cruise.

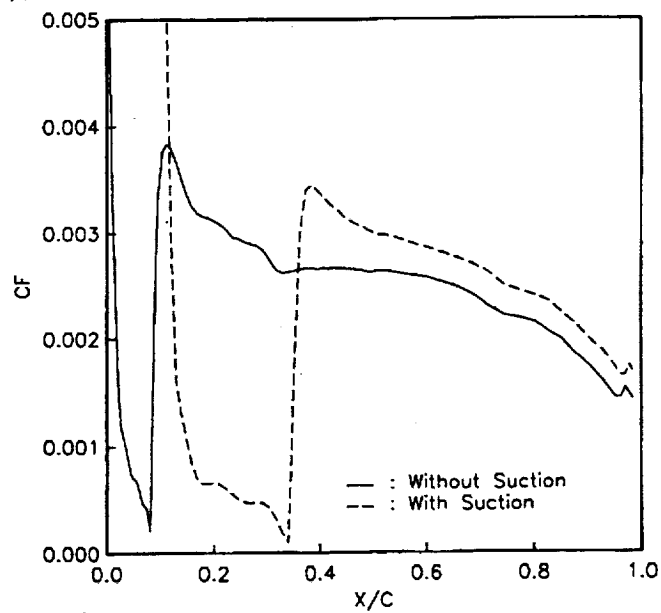


Figure 8.—Laminar/Turbulent transition at cruise.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE January 1994	3. REPORT TYPE AND DATES COVERED Technical Memorandum		
4. TITLE AND SUBTITLE Flow Analysis for the Nacelle of an Advanced Ducted Propeller at High Angle-of-Attack and at Cruise with Boundary Layer Control		5. FUNDING NUMBERS  WU-535-03-10		
6. AUTHOR(S)  D.P. Hwang, D.R. Boldman, and C.E. Hughes				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)  National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		8. PERFORMING ORGANIZATION REPORT NUMBER  E-8181		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)  National Aeronautics and Space Administration Washington, D.C. 20546-0001		10. SPONSORING/MONITORING AGENCY REPORT NUMBER  NASA TM-106371 AIAA-94-0391		
11. SUPPLEMENTARY NOTES  Prepared for the 32nd Aerospace Sciences Meeting and Exhibit sponsored by the American Institute of Aeronautics and Astronautics, Reno, Nevada, January 10-13, 1994. Responsible person, D.P. Hwang, (216) 433-2187.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT  Unclassified - Unlimited Subject Category 02		12b. DISTRIBUTION CODE		
13. ABSTRACT (Maximum 200 words)  An axisymmetric panel code and a three-dimensional Navier-Stokes code (used as an inviscid Euler code) were verified for low speed, high angle-of-attack flow conditions. A three-dimensional Navier-Stokes code (used as an inviscid code), and an axisymmetric Navier-Stokes code (used as both viscous and inviscid code) were also assessed for high Mach number cruise conditions. The boundary layer calculations were made by using the results from the panel code or Euler calculation. The panel method can predict the internal surface pressure distributions very well if no shock exists. However, only Euler and Navier-Stokes calculations can provide a good prediction of the surface static pressure distribution including the pressure rise across the shock. Because of the high CPU time required for a three-dimensional Navier-Stokes calculation, only the axisymmetric Navier-Stokes calculation was considered at cruise conditions. The use of suction and tangential blowing boundary layer control to eliminate the flow separation on the internal surface was demonstrated for low free stream Mach number and high angle-of-attack cases. The calculation also shows that transition from laminar flow to turbulent flow on the external cowl surface can be delayed by using suction boundary layer control at cruise flow conditions. The results were compared with experimental data where possible.				
14. SUBJECT TERMS  Nacelle; High angle of attack; Cruise; Panel code; Euler; Navier-Stokes, Boundary layer control			15. NUMBER OF PAGES 8	
			16. PRICE CODE A02	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	