TWENTYFIFTH EUROPEAN ROTORCRAFT FORUM

Paper n° C12

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS TO THE DESIGN OF THE BA 609

BY

J. C. NARRAMORE, D. A. PLATZ, A. G. BRAND

BELL HELICOPTER TEXTRON INC., FORT WORTH, TEXAS, USA

SEPTEMBER 14–16, 1999
ROME
ITALY

ASSOCIAZIONE INDUSTRIE PER L'AEROSPAZIO, I SISTEMI E LA DIFESA
ASSOCIAZIONE ITALIANA DI AERONAUTICA ED ASTRONAUTICA

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS TO THE DESIGN OF THE BA 609

J. C. Narramore
D. A. Platz
A. G. Brand
Bell Helicopter Textron, Inc.
Fort Worth, Texas, U.S.A

1. ABSTRACT

Cost and weight considerations factor heavily into the design of the BA 609 civil tiltrotor. Therefore, aerodynamic design solutions must be developed in a timely manner within stringent constraints. Computational, empirical, and wind tunnel testing tools are being used to produce optimal aerodynamic solutions that satisfy these constraints. Because of the detailed results that are available from computational fluid dynamics (CFD) methods, these tools are being used to produce new aerodynamic design solutions that address complex flow issues. This paper describes some of the successes obtained by using computational fluid dynamic methods during the design of the BA 609 Civil Tiltrotor.

2. INTRODUCTION

To predict aerodynamic characteristics and air-loads, Bell Helicopter has pursued several approaches including wind tunnel testing, empirical methods, analytical methods, and computational fluid dynamics (CFD) methods. The goal of constantly improving the design process has led to the development of many new techniques and upgrades in aerodynamic design methodology. The advent of computational methods for aerodynamic design and analysis for rotorcraft vehicles provides new opportunities to improve the design approach. As these methods are developed, computational results are continually compared to wind tunnel test data to validate the accuracy of the computations (Ref. 1).

Developments in high-speed computers have also had an impact on the time required to perform aerodynamic design and the types of problems that may be solved. As computers have become more capable, solutions with increasing detail have been produced. In addition, improved modeling of the physics of the flow and of the configuration geometry is also made possible. Interactive graphic workstations have allowed a new approach to the design of aerodynamic components and the development of computational grids for complex shapes. Supercomputers have generated solutions to problems that heretofore would have taken too long to be of practical use in the design process.

The development of CFD methodology at Bell Helicopter Textron, Inc. was initiated in the early 1970's with two-dimensional (2–D) airfoil analysis and design methods. This included both incompressible and transonic full-potential methods. In the early 1980's, three-dimensional (3–D) potential flow solutions became the standard for distributing external airloads on rotorcraft. In 1987, evaluations of transonic full-potential codes for rotor blades were conducted. Navier-Stokes methods were being used for some practical rotorcraft component problems in 1991. By 1995, full rotorcraft configurations were being analyzed using Navier-Stokes codes. This growth in capability has paralleled advances in computer hardware and the development of tools that improve the productivity for CFD methods. As a result of this growth in capability, state-of-the-art methods were in place to aid in the design of the BA 609.

3. CFD TOOLS

Several aerodynamic computational tools have been used during the design of the BA 609. These tools are used for grid generation, Navier-Stokes analysis, surface manipulation, and inverse airfoil design.

3.1 Grid Generation

Surface grids are generated from point distributions extracted from a CATIA surface definition. For the 3-D aircraft models, surface grids for the individual components are generated with proper overlap at the intersections using the GRIDGEN (Ref. 2) code. A body conforming volume grid for each surface component is produced using a hyperbolic grid generator code, HYPGEN (Ref. 3). This code generates highly clustered and smoothly varying grids by solving a hyperbolic differential equation. A graphical user interface simplifies input and execution and a check for negative cell volumes is conducted while executing. The PEGSUS (Ref. 4) code is used to develop the hole cutting and connectivity stencils for the component grids.

3.2 Navier-Stokes Code

The primary CFD analysis tool being used at Bell Helicopter is the OVERFLOW code (Ref. 5) developed by Pieter Buning at NASA. OVERFLOW is a Navier-Stokes code with various boundary conditions and turbulence models available. A primary advantage of this code is its ability to solve problems using overset grids. This feature allows complex geometries to be modeled to any desired level of accuracy. For the cases used in the BA 609 studies, OVERFLOW was run using the Spalart-Allmaras turbulence model (fully turbulent), central-differencing of the spatial terms, and the three-factor diagonal implicit scheme.

3.3 Post Processing

FOMOCO (Ref. 6) is used to calculate the forces and moments from the OVERFLOW solution. Since OVERFLOW uses overset grids, the overlapped regions of the surface grids must be eliminated and connectivity established between adjacent grids before the surface pressures are integrated. The FOMOCO code performs these operations. Visualization tools are used to interpret the computational results. These include the FAST code (Ref. 7), which reads OVERFLOW-generated files and the Bell ADAM post processor that visualizes surface pressures, integrates forces, and distributes airloads to NASTRAN models.

3.4 Inverse Design Tools

Inverse tools currently being used include conformal mapping methods for single element airfoils and predictor/corrector techniques for multi-element and transonic airfoil design (Ref. 1). Inverse airfoil design methods are advantageous over trial-and-error perturbation methods since the pressure distribution, which determines the boundary layer development and the lift (moment) of the section, is specified *a priori*. Thus, the performance is specified and the coordinates that will produce that performance are determined. If geometric constraints are critical, inverse design methods can be used to determine the best aerodynamic configuration with the given constraints. Since inverse conformal methods are extremely fast (Ref. 1), these design methods are nearly instantaneous on a modern graphic workstation. As soon as a selected design parameter is adjusted, the result appears on the screen. The ability to work interactively on a graphic workstation greatly improves productivity for airfoil design.

4. APPLICATION OF CFD

During the design of the BA 609, CFD methods were used to develop performance enhancing surface contours. CFD was also used as a tool to generate airload distributions for structural design. The CFD methods were used to develop new inboard rotor and empennage airfoils, to improve the wing-to-fuselage fairing, to reduce the size and lower the drag of the tail cone, to determine inlet and exhaust locations, and to provide airload distributions for the structural design of landing gear doors, wing fairings, spinners, nacelles, over-wing fairings, conversion actuator fairings, wing skins, tail surfaces, and fuselage.

4.1 Development of New Inboard Rotor Airfoils

To minimize costs and risks, the V-22 rotor airfoils were also used on the BA 609 rotor. However, since the flex beam motions inside the blade cuff do not scale directly and reduced steady pitch link loads were desired, the inboard blade airfoils required redesign. The goal was to reduce the pitching moment in the inboard region by 60% while maintaining the V-22 lift characteristics and minimizing the drag. The approach taken was to use very fast inverse conformal mapping techniques to generate airfoils with desired moment characteristics that satisfy the clearance constraints. The trailing edge was then blunted by fitting a straight line to the airfoil surface aft of 98% chord. The resulting airfoil was analyzed using potential flow with boundary layer methods to determine if the performance of the new section was acceptable. Finally, OVERFLOW Navier-Stokes analysis of the new design shapes and the baseline V-22 airfoils was conducted to make the final section selection.

Fig. 1 illustrates the user interface for an inverse conformal mapping solution that was generated. The upper left window depicts the input pressure distribution and the lower left window shows the resulting airfoil and clearance constraint. The panel on the right shows the input parameters that define the specified pressure distribution. Since this method is a conformal mapping, the trailing edge thickness of the airfoil is zero. Input velocity parameters were adjusted to provide a thick trailing edge when a linear fit to the trailing edge is computed at 98% chord. Using the inverse methods, a family of airfoils with different trailing edge thicknesses was created, all satisfying the clearance constraint. After defining the geometry, a Navier-Stokes trade study was conducted to determine the trailing edge thickness required to satisfy the maximum lift design requirement. Fig. 2 shows a typical grid distribution about a candidate inboard region airfoil. Results indicate that as the trailing edge thickness is increased, the maximum lift capability is increased; however, the drag level also increases. The trailing edge thickness for the final airfoil was chosen to maintain the maximum lift capability of the corresponding V-22 airfoil section. In Fig. 3, the resulting BA 609 28% thick airfoil is compared to the V-22 28% airfoil. Fig. 4 shows that the BA 609 airfoil lift curve slope is steeper than the V-22 airfoil, and that the maximum lift coefficient is matched by the final BA 609 airfoil shape. In Fig. 5, the drag curve of the BA 609 airfoil is compared to the V-22 airfoil. The figure indicates that minimum drag is equivalent and the low drag region is much wider for the BA 609 section. The pitching moment of the two airfoils is compared in Fig. 6. Results show that the pitching moment at zero lift coefficient was reduced by 66% with the BA 609 cuff airfoil.

As a result of this procedure, new inboard BA 609 airfoils were developed that provide improvements in performance compared to using scaled V-22 airfoils. In addition, because of the reduction in pitching moment, the pitch link loads were also reduced.

4.2 Wing-to-fuselage Fairing Design

With a three-dimensional CFD solution, we can examine the flow in detail at any location on the aircraft. In

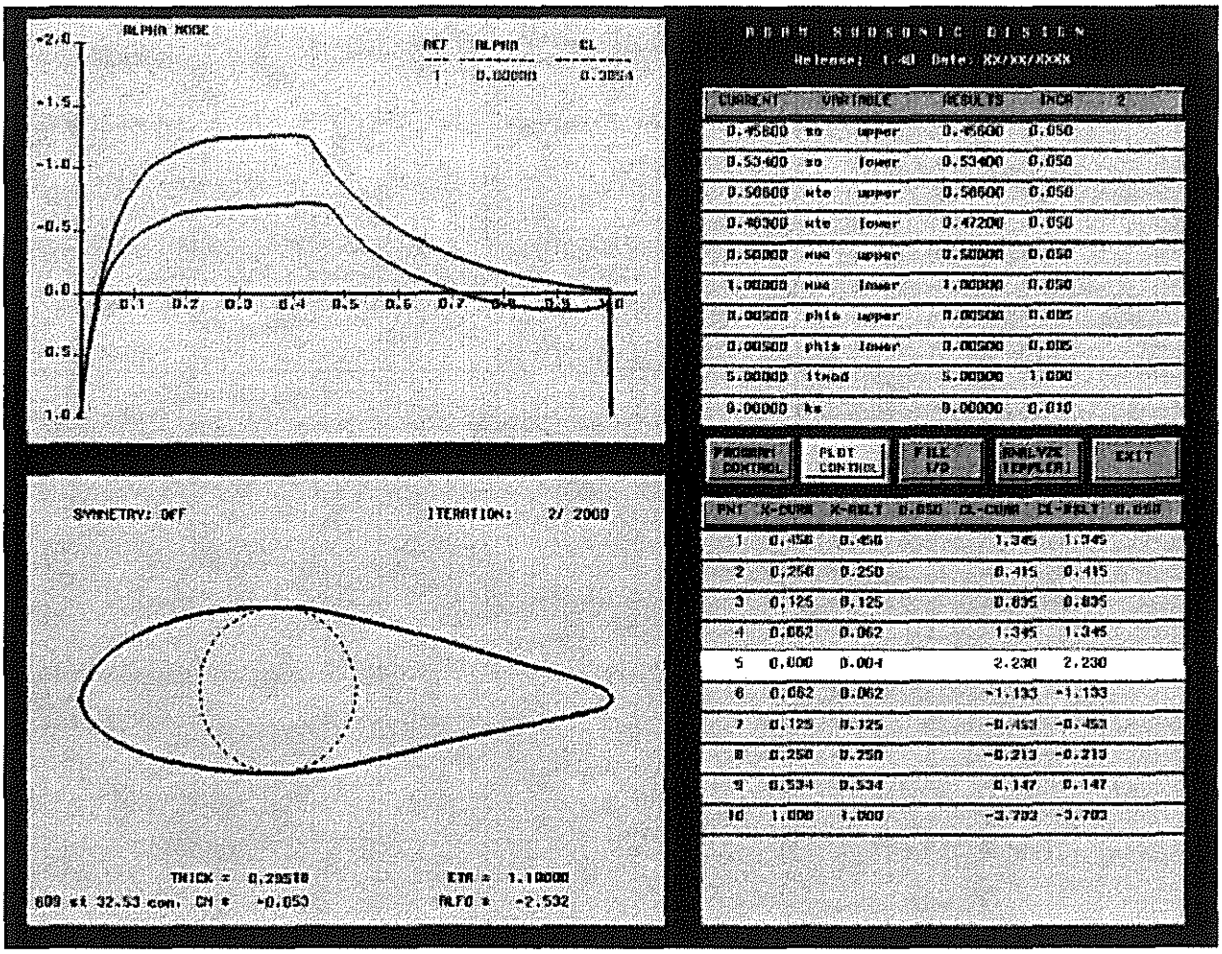

Fig. 1. User graphical interface for subsonic inverse airfoil design method.

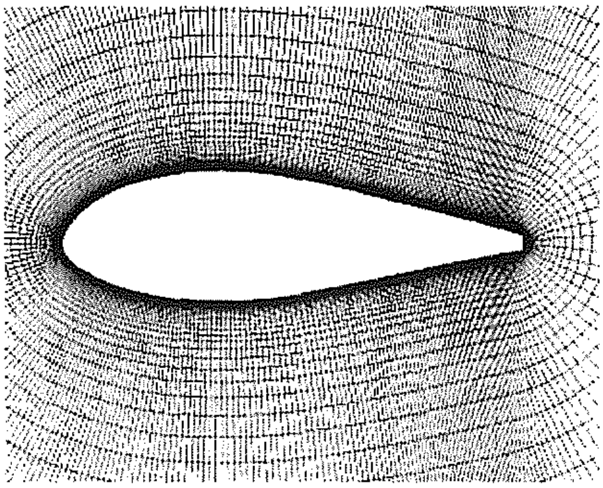

Fig. 2. Typical grid for Navier-Stokes analysis is 301 × 101 × 3 nodes.

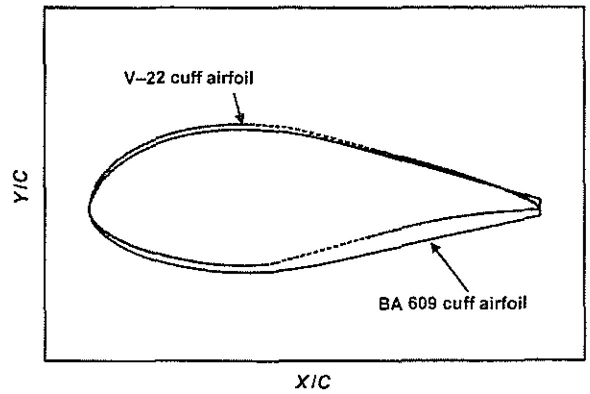

Fig. 3. The BA 609 cuff airfoil has less camber and more trailing edge thickness.

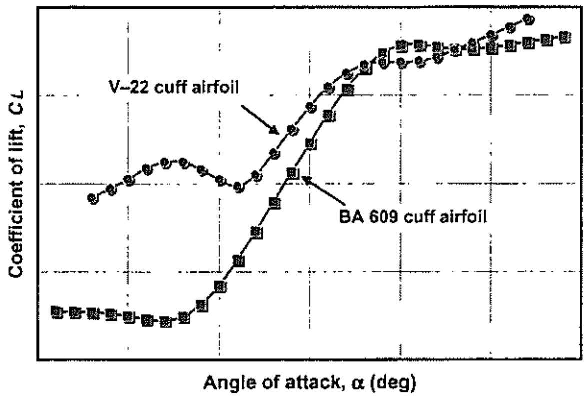

Fig. 4. The BA 609 cuff airfoil improves the lift coefficient versus angle of attack.

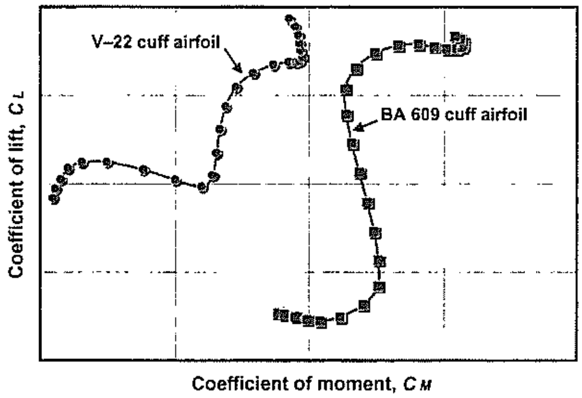

Fig. 6. The BA 609 cuff airfoil pitching moment is reduced by 66%.

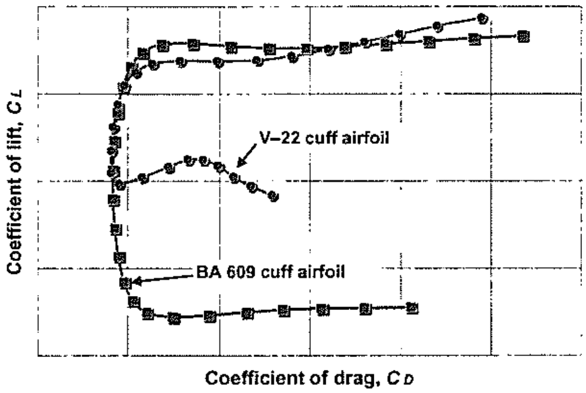

Fig. 5. The BA 609 cuff airfoil drag curve is much wider than for the V–22 cuff airfoil.

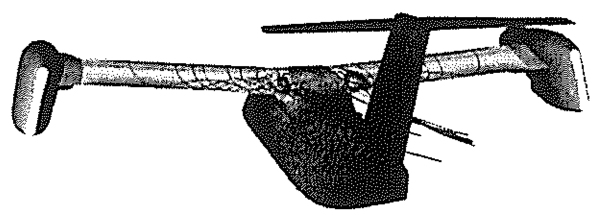

Fig. 7. Computed flow about the baseline BA 609 at wind tunnel conditions.

this capacity, CFD technology provides an unmatched diagnostic capability for rotorcraft design. In the BA 609, CFD was used to determine the cause of an aerodynamic issue found in a wind tunnel test and a CFD-based design solution was implemented to correct that issue.

Preliminary BA 609 tests, conducted at the Texas A&M University 7- × 10-ft (2- × 3-m) wind tunnel, found that the vertical tail effectiveness was reduced at high angles of attack. This situation could lead to decreased directional stability of the aircraft and was undesirable.

To investigate this situation, the BA 609 was modeled using computational fluid dynamic methods while the wind tunnel test was still in progress. A three-dimensional model had been generated to represent the baseline configuration. The OVERFLOW code generated results for angles of attack where the reduced tail effectiveness was observed in the wind tunnel test. These runs were performed on a Cray supercomputer at NASA Ames Research Center. A typical solution for this three-dimensional model of the BA 609 required approximately 15 hours of CPU time on a Cray C90. Fig. 7 shows the computed flow for the baseline BA 609 model at the wind tunnel model conditions. The solution indicated a flow separation in the area where the wing joins the body at this high angle of attack. The separated flow creates a large region of turbulence, which moves aft along the fuselage and impinges on the vertical tail. Based on evaluations of these computational results, a modification to the wing/fuselage fairing was pursued to improve the effectiveness of the vertical tail at high angles of attack.

A project was initiated to develop a new wing/fuselage fairing using the computational methods to generate results. The objective of this computational design effort was to eliminate the computed flow separation at the wing-to-fuselage junction. New fairing shapes were created by adjusting the surface grids at the wing/fuselage intersection and regenerating the volume grid and holes for these grids. Two iterations were required to produce a configuration that produced substantially improved computed flow. These geometry iterations and the generation of computational results for the new designs required a total of three weeks for completion. Fig. 8 shows a comparison of the computed results for the modified fairing and the original baseline design. It shows that the new fairing significantly reduces the flow separation at the wing-to-fuselage junction. Based on these results it was felt that the BA 609 configuration could be improved with the CFD-developed fairing.

The surface coordinates of the new design were transferred to CATIA and stereolithography was used to

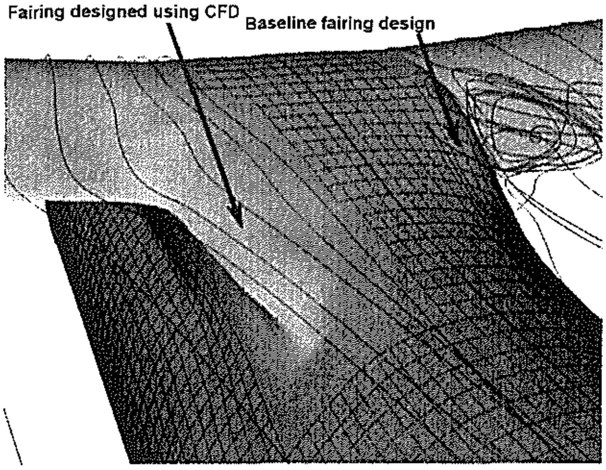

Fig. 8. The new BA 609 at wing/fuselage fairing significantly reduces flow separation.

Fig. 9. Wind tunnel testing of the new fairing design produced improved performance.

generate a part for the wind tunnel model shown in Fig. 9. That part was sent to the wind tunnel in time to be incorporated into the model while the test was still in progress.

Wind tunnel testing with the new fairing showed a 16% improvement in the directional stability and a 0.18 ft^2 (0.0167 m^2) reduction in the drag. Since no parts had to be fabricated for the computational analysis, CFD was able to provide a low-cost alternative to conventional trial and error wind tunnel testing methods and provided significant insight into the cause of reduced vertical tail effectiveness.

4.3 Midwing Exhaust Pressures

In order to cool equipment in the midwing area, inlets and exhausts for cooling flow are necessary. Initial evaluations of the size and placement of the inlet and exhaust were based on a panel method solution (Ref. 8). When final sizing and placement studies were conducted, the OVERFLOW code was used to provide additional midwing fairing pressure information. At low angles of attack, the panel method and Navier-Stokes solutions were equivalent, as is shown in Fig. 10. In this figure, the pressures aft of the wing on the BA 609 from the panel code and OVERFLOW are given. The pressure coefficients shown in the left of the figure are at the station shown by the plane in the right of the figure. As can be seen, at an angle of attack of zero degrees, the panel method solution is very similar to the Navier-Stokes solution. This indicates that the flow is smooth and that there is no flow separation on the fuselage at this cruise condition. However, at very high angles of attack, the boundary layer thickens and the pressures from the panel method produce results that are different from the Navier-Stokes results (Fig. 11). The recovery of pressure aft of the wing is not as positive for the Navier-Stokes solution as the panel method would indicate. Because the Navier-Stokes code models the thickening of the boundary layer and flow separation, it was used for the placement and sizing determination of the midwing fairings for the BA 609. Using the Navier-Stokes results ensures that adequate cooling flow is obtained at all flight conditions.

4.4 Distribution of Airloads

Computational fluid dynamic results are also being used to calculate airload distributions on the BA 609. During the design of external components, surface pressures are needed to establish the airload contribution to the total load. Critical load conditions are established for each external component by considering combinations of maximum local flow velocity, extremes in local flow direction, and Mach number as determined by the design flight envelope and maneuvering requirements. CFD analysis is run at the selected critical flight conditions to produce a detailed distribution of pressures on the surface of the model. The Navier-Stokes grid model that was used for these runs is represented in Fig. 7. Therefore, the approach was efficient in that only new cases were run for the airload distributions on the existing grid model.

For the external structural components, the CFD pressure results for each case are mapped onto a NASTRAN model using a method developed at Bell. Fig. 12 shows a NASTRAN surface geometry for the BA 609 nacelle. Pressures from an OVERFLOW run have been mapped onto this geometry. The surface pressures are depicted by colors in this figure. The dark regions depict suction pressures and the lighter regions depict positive pressures. As part of this process, input files are generated that represent the distributed pressure on the NASTRAN model. This deck is transferred to the structural loads groups to provide air load information for the design of the structure. For the BA 609, this allowed fast generation of critical air loads and incorporation of this CFD information into the structural design codes.

5. SUMMARY

Computational fluid dynamics is becoming an integral part of the aircraft design process. Navier-Stokes codes allow details to be modeled and new designs to be produced that improve the aerodynamic characteristics. In addition, methods have been developed that allow the CFD air load distributions to be mapped to the

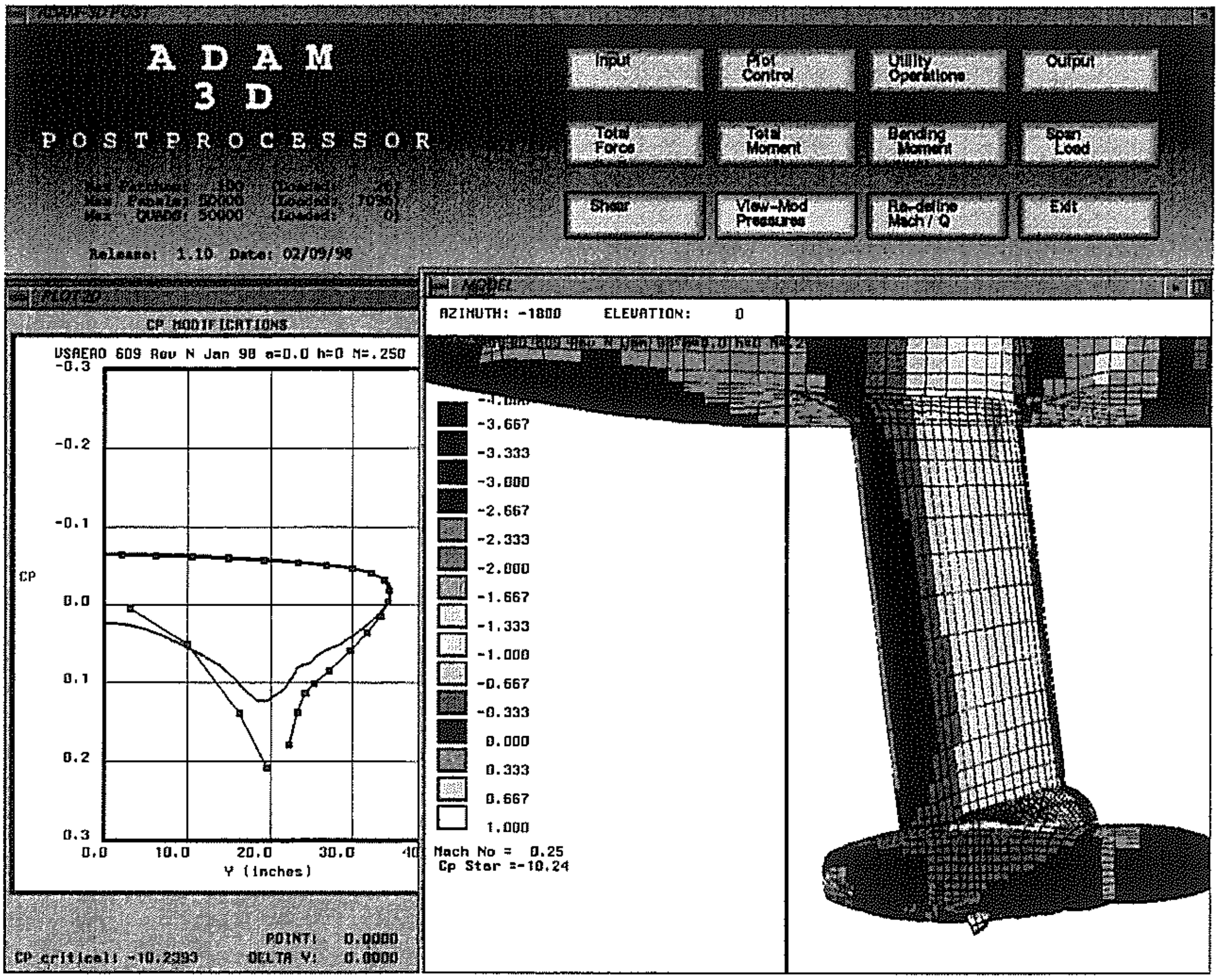

Fig. 10. Panel method and Navier-Stokes pressures are similar at low angles of attack.

NASTRAN structural design code. As a result of the application of CFD to the design of the BA 609, new design concepts are being developed that improve the performance, lower the operating cost, and provide the minimum weight solution.

6. ACKNOWLEDGEMENTS

The authors wish to express their appreciation to Ashok Agnohotri, Ross Menger, Bob Milliken, and Ted Trept for valuable discussions during the preparation of this paper, Don Axley for technical assistance, and Chuck Gatlin who finalized the graphics and layout.

7. REFERENCES

1. J. C. Narramore, "Computational Fluid Dynamics Development and Validation at Bell Helicopter," AGARD Conference Proceedings No. 552, "Aerodynamics and Aeroacoustics of Rotorcraft", Paper No. 15, August 1996.
2. J. P. Steinbrenner and J. R. Chawner, "The GRIDGEN Version 9 Multiple Block Grid Generation Software," MDA Engineering Report 94-01, August 12, 1994.
3. W. M. Chan, I. T. Chiu, and P. G. Buning, "User's Manual for the HYPGEN Hyperbolic Grid Generator and the HGUI Graphical User Interface," NASA TM-108791, October 1993.
4. N. E. Suhs and R. W. Tramel, "PEGSUS 4.0 User's Manual," AEDC-TR-91-8, AEDC/PA, Arnold Air Force Base, Tenn., 1991.
5. P. G. Buning, D. C. Jespersen, T. H. Pulliam, W. M. Chan, J. P. Slotnick, S. E. Krist, and K. J. Renze, "OVERFLOW User's Manual Version 1.7v," NASA Langley Research Center, Hampton, VA, 1997.
6. W. M. Chan and P. G. Buning, "User's Manual for FOMOCO Utilities - Force and Moment Computation Tools for Overset Grids," NASA TM-110408, July 1996.
7. Val Watson, "Software for Visualization of Fluid Dynamics," presented at the Computational Fluid Dynamics Conference, NASA Ames Research Center, March 12-14, 1991.
8. B. Maskew, "VSAERO User's Manual," NASA CR 166-476, November 1982.

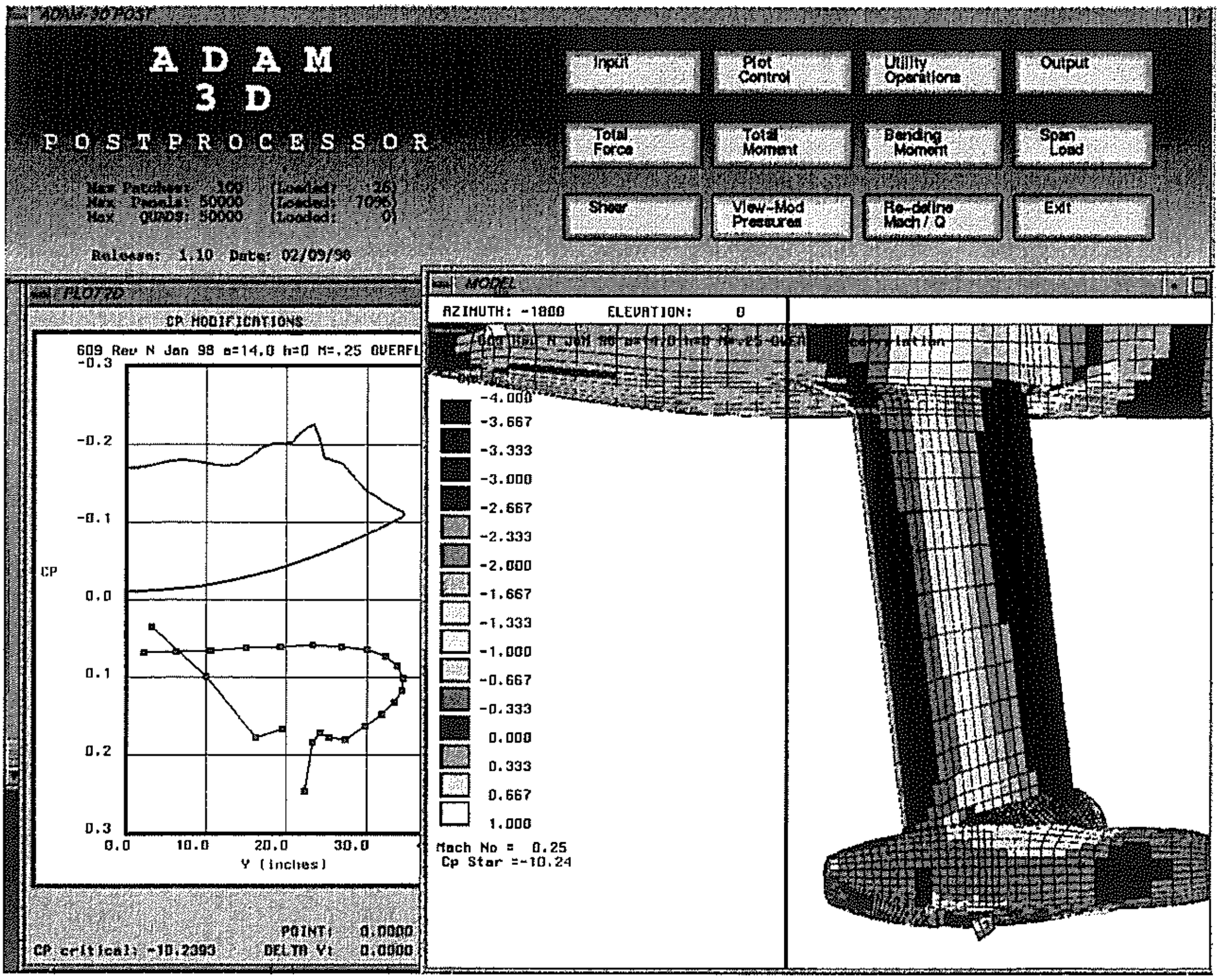

Fig. 11. At very high angles of attack, Navier-Stokes improves the pressure prediction.

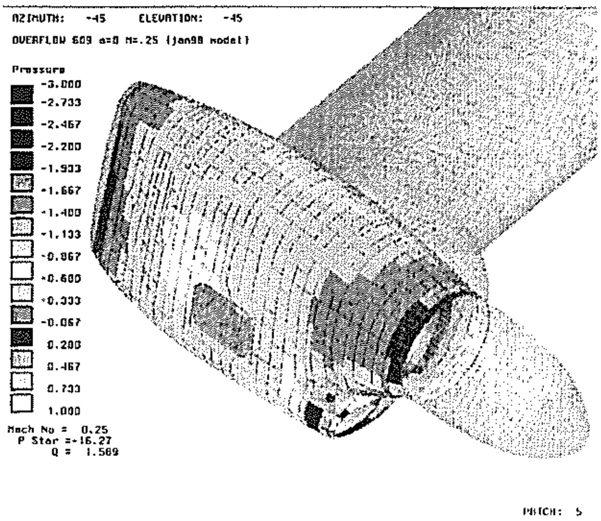

Fig. 12. CFD pressures were mapped onto the NASTRAN surface model.