A PROCEDURE FOR AUTOMATING CFD SIMULATIONS
OF AN INLET-BLEED PROBLEM

Wei J. Chyu[*]
National Aeronautics and Space Administration
Ames Research Center
Moffett Field, California 94035

Mark J. Rimlinger[¶] and Tom I-P. Shih[§]
Carnegie Mellon University
Pittsburgh, Pennsylvania 15213

SUMMARY

A procedure was developed to improve the turn-around time for computational fluid dynamics (CFD) simulations of an inlet-bleed problem involving oblique shock-wave/boundary-layer interactions on a flat plate with bleed into a plenum through one or more circular holes. This procedure is embodied in a preprocessor called AUTOMAT. With AUTOMAT, once data for the geometry and flow conditions have been specified (either interactively or via a namelist), it will automatically generate all input files needed to perform a three-dimensional Navier-Stokes simulation of the prescribed inlet-bleed problem by using the PEGASUS and OVERFLOW codes. The input files automatically generated by AUTOMAT include those for the grid system and those for the initial and boundary conditions. The grid systems automatically generated by AUTOMAT are multi-block structured grids of the overlapping type. Results obtained by using AUTOMAT are presented to illustrate its capability.

INTRODUCTION

Effective control of shock-wave induced boundary-layer separation by using bleed is vital to the successful operation of inlets in supersonic aircraft. This is because the strong external/internal shock waves generated in a mixed compression inlet often induce boundary-layer separations that can cause the potentially dangerous unstart to occur. The importance of bleed in controlling shock-wave/boundary-layer interactions has led to a number of experimental and computational studies. Although all experimental studies showed that bleed can control shock-wave induced flow separation, they failed to show how bleed-hole geometry and flow conditions about the bleed holes influence the effectiveness of the bleed process. Computational studies of shock-wave/boundary-layer interactions with bleed fall into two groups. The first group modelled the bleed process by using boundary conditions which treated bleed holes collectively as a slot or as a porous surface without resolving the flow through the bleed holes (refs. 1 - 6). The second group studied the bleed process in detail by resolving the flow through each bleed hole (refs. 7 - 16). The second group, in early stages of their work, performed two-dimensional (2-D) studies in which each bleed hole is modelled as a slot on a flat plate (refs. 7 - 12). Later, three-dimensional (3-D) studies were performed where air in the boundary layer above a plate was bled through circular holes into a plenum (refs. 13 - 16). These studies showed the complex nature of shock-wave/boundary-layer interactions with bleed. These studies also showed that the bleed process is sensitive to many design and operating parameters with different ones dominating under different conditions.

Since a very large number of design and operating parameters need to be investigated for the inlet-bleed problem, computational fluid dynamics (CFD) simulation is an attractive approach to get the required information. But, in order for CFD to be useful and cost-effective, the turn-around time must be good in addition to obtaining meaningful results. There are several factors which affect the turn-around time of CFD simulations. Of these, grid generation is the most critical. For complicated geometries, the generation of good quality grids require not only substantial time, but also considerable expertise from the user. Thus, an automatic grid generator would greatly reduce turn-around time. To further reduce turn-around time, input

---

[*] Research Scientist, High-Speed Aerodynamics Branch, Applied Aerodynamics Division.
[¶] Graduate Student, Department of Mechanical Engineering.
[§] Professor, Department of Mechanical Engineering.

files that contain initial and boundary conditions as well as other information needed by the CFD code in order to generate solutions on the computer also should be generated automatically.

In this study, a CFD preprocessor, called AUTOMAT, was developed to reduce turn-around time for CFD simulations of an inlet-bleed problem. With AUTOMAT, once data for geometry and operating conditions have been specified (either interactively or via a namelist), it will automatically generate all input files needed to simulate the prescribed inlet-bleed problem by using the PEGASUS (refs. 17 and 18) and OVERFLOW (ref. 19) codes. The input files generated by AUTOMAT include those for the grid system as well as initial and boundary conditions. The inlet-bleed problem setup by AUTOMAT is one in which a number of geometric and operating parameters of importance to realistic inlet-bleed systems in supersonic aircraft can be investigated.

The organization of the rest of this paper is as follows. First, description is given on the inlet-bleed problem setup by AUTOMAT, including the geometric and operating parameters that can be varied for parametric studies. Next, the formulation and the numerical method of solution are outlined. This is then followed by a description of AUTOMAT, its operation and the built-in knowledge-based system. This paper concludes with some sample inlet-bleed problems setup by AUTOMAT along with some sample solutions for the flow field. A description of all input parameters into AUTOMAT is given in the Appendix.

DESCRIPTION OF THE INLET-BLEED PROBLEM

The inlet-bleed problem of interest in this study involves a subsonic or supersonic turbulent boundary layer flowing past a flat plate, bleed of the boundary layer through one or more circular holes which can be normal or inclined and arranged in different patterns, and a plenum where the bleed is discharged. Figure 1 shows a typical example of such an inlet-bleed problem in which air above the flat plate is bled through four rows of inclined holes arranged in a staggered fashion. For this problem, the domain is the region bounded by the dashed lines which includes the region above the flat plate, the plenum, and a number of "half" bleed holes in the case of multiple rows of holes. Only "half" bleed holes needed to be included in the domain because of the symmetry in the spanwise direction.

The aforementioned inlet-bleed problem is selected for study because it enables the investigation of some of the most important design and operating parameters relevant to inlet-bleed systems in supersonic aircraft. More specifically, the following geometric parameters can be investigated (see Fig. 1):

- bleed-hole pattern (either one hole, three-holes in line, or rows of holes arranged in a staggered fashion),
- number of rows if staggered arrangement (N),
- spacing in streamwise direction (L_x, measured between centers of holes),
- spacing in spanwise Direction (L_y, measured between centers of holes),
- diameter of bleed holes (D),
- orientation or inclination of bleed holes (α),
- thickness of flat plate (L_t), and
- plenum dimensions (L_{P1}, L_{P2}).

The operating parameters that can be investigated by this inlet-bleed problem include (see Fig. 1):

- with or without incident shock wave impinging on the boundary-layer next to flat plate,
- shock-wave-generator half angle (β)[*],
- inviscid-shock-impingement point on flat plate (measured from the center of holes in the first row),
- freestream Mach number (M_∞),
- freestream static pressure (P_∞),
- freestream static temperature (T_∞),
- adiabatic or isothermal wall,

---

[*]   Note that the shock-generator half angle and the freestream Mach number must be such that the incident and reflected shock waves can be analyzed as a one-dimensional problem. Very weak oblique shocks become two-dimensional when approaching a flat plate.

- boundary-layer momentum thickness (θ), and
- plenum back pressure (P_b).

FORMULATION AND NUMERICAL METHOD OF SOLUTION

The inlet-bleed problem described in the previous section was modelled by the density-weighted, ensemble-averaged conservation equations of mass, momentum ("full compressible" Navier-Stokes), and total energy written in generalized coordinates and cast in strong conservation-law form. The effects of turbulence were modelled by the Baldwin-Lomax algebraic turbulence model (ref. 20).

In order to obtain solutions to the conservation equations, boundary and initial conditions are needed. The boundary conditions (BCs) setup by AUTOMAT for the different boundaries shown in Fig. 1 are as follows. At the inflow boundary where the flow is supersonic everywhere except for a very small region next to the flat plate, two types of BCs are imposed. Along segment A-B, all flow variables are specified at the freestream conditions except for the streamwise velocity (which has a turbulent boundary-layer next to the flat plate) and the stagnation temperature (which is kept constant everywhere including the boundary layer so that the static temperature varies in the boundary layer). The velocity profile in the turbulent boundary layer is described by using the method of Huang, et al. (ref. 21). Along segment B-C, post-shock conditions based on inviscid, oblique, shock-wave theory are specified. These post-shock conditions are also specified along the freestream boundary (segment C-D). At the outflow boundary where the reflected shock wave exit the computational domain, the flow is also mostly supersonic except for a small region next to the flat plate so that all flow variables are extrapolated. Here, linear extrapolation based on three-point, backward differencing was employed. When there are symmetry boundaries as shown in Fig. 1, the BCs imposed are zero derivatives of the dependent variables except for the velocity component normal to those boundaries which was set equal to zero. At the exit of the plenum where the flow is subsonic, a back pressure (P_b) is imposed, and density and velocity are extrapolated in the same manner as the variables at the outflow boundary. At all solid surfaces, the no-slip condition and adiabatic walls are imposed.

Even though only steady-state solutions are of interest, initial conditions are needed because the unsteady form of the conservation equations are used. The initial conditions employed in this study are as follows. In the region above the flat plate, the initial condition is the two-dimensional, steady-state solution for an incident and a reflected oblique shock wave on a flat plate based on inviscid, oblique, shock-wave theory. The streamwise velocity profile, however, is modified according to the method of Huang, et al. (ref. 21). This necessitated the density and static temperature to be modified as well in order to maintain constant stagnation temperature in the boundary layer. The initial conditions used in the bleed hole and plenum are stagnant air at constant stagnation temperature (T_o) and static pressure (P_b).

Solutions to the ensemble-averaged conservation equations along with the corresponding initial and boundary conditions are obtained by using the OVERFLOW code (ref. 19). The OVERFLOW code contains many algorithms. The one used in this study is as follows: Inviscid flux-vector terms in the ξ-direction are upwind differenced by using the flux-vector splitting procedure of Steger and Warming (ref. 22). Inviscid flux-vector terms in directions normal to the ξ-direction are centrally differenced in order to reduce artificial dissipation in those direction. Diffusion terms in all directions are also centrally differenced. The time-derivative terms are approximated by the Euler implicit formula since only steady-state or quasi-steady-state solutions are sought. The system of nonlinear equations that resulted from the aforementioned approximations to the space- and time-derivatives are analyzed by using the partially split method of Steger, et al. (ref. 23). In OVERFLOW, Jacobians and metric coefficients are interpreted as grid-cell volumes and grid-cell surface areas, respectively. In this regard, all algorithms in OVERFLOW are implemented in the finite-volume manner. However, BCs in OVERFLOW are implemented in a finite-difference manner in order to enhance flexibility and ease in investigating different problems. Grid systems that are needed to obtain solutions are described in the next section.

THE PREPROCESSOR AUTOMAT

For the inlet-bleed problem shown in Fig. 1 and described in the previous two sections, once data describing the geometry and operating conditions have been specified, the preprocessor, AUTOMAT, will automatically generate all input files needed by PEGASUS and OVERFLOW to perform CFD simulations. The data input into AUTOMAT can be interactive or non-interactive. If it is interactive, then it relies on

PLOT3D (ref. 24) for the graphics output on the screen. Thus, PLOT3D must be available on whatever computer being used if interactive mode is desired. Whether the data input is interactive or non-interactive, outputs of AUTOMAT are as follows:

- a grid system in PLOT3D format for input to OVERFLOW (grid.in),
- an intial condition file in PLOT3D format for input to OVERFLOW (q.save),
- a data file for input to OVERFLOW (bleed.inp), and
- an input file for PEGASUS to create the fort.2 file needed by OVERFLOW for overlapping grids.

In the next section, how AUTOMAT interfaces with the user in an interactive session is described (see APPENDIX for a non-interactive session). Afterwards, the knowledged-based system built into AUTOMAT to generate grid systems is briefly outlined.

User Interface

AUTOMAT is written in Fortan 77. Once installed on a computer, AUTOMAT is invoked by entering "automat" and then pressing return or enter. On computers such as an Iris workstation with PLOT3D available, the interface with the user can be interactive or non-interactive. If PLOT3D is not available, then only non-interactive sessions are possible. For interactive sessions, the user will be prompted on the screen for the input data. For non-interactive sessions, the inputs are all contained in a namelist file, which can be modified by the user via any editor (e.g., vi in unix). Here, it is noted that the namelist file required for non-interactive sessions is automatically generated after every interactive session to minimize effort needed to input data for future sessions.

A typical interactive session with AUTOMAT proceeds as follows:
>Enter "automat" to invoke program.
>interactive or non-interactive?
    Enter y or yes for interactive, and n or no for non-interactive.
    If interactive, then all inputs would be prompted for on the screen.
    If non-interactive, then all inputs can be placed in a namelist file. After each interactive session, a namelist file is created automatically to facilitate future sessions.
>units (English or metric)?
    Enter e for English, and m for metric.
>Requests for input on operating parameters one at a time (freestream Mach number, freestream static pressure, freestream static temperature, gas constant (universal gas constant / gas molecular weight), Reynolds number per unit length (based on freestream density, velocity, and viscosity), boundary-layer momentum thickness, with or without incident shock (y for shock, n for no shock), shock-generator half angle (only asked if there is an incident shock), plenum back pressure).
>Requests for verification on inputted operating parameters. Enter y or yes to accept inputs. Enter n or no to modify inputs.
>Requests for inputs on geometric parameters one at a time (bleed-hole pattern (either one hole or rows of holes arranged in staggered fashion), bleed-hole diameter, bleed-hole inclination, number of rows, spacing between hole centers in streamwise direction, spacing between hole centers in spanwise direction, plate thickness, shock impingement location on flat plate under inviscid condition (measured from the center of holes in the first row), plenum dimensions).
>Requests for verification on inputted geometric parameters. Enter y or yes to accept inputs. Enter n or no to modify inputs.
>Shows grid generated in bleed hole on workstation.
>Requests for acceptance. Enter y or yes to accept grid generated. Enter n or no to modify grid generated.
>Shows grid generated above flat plate on workstation.
>Requests for acceptance. Enter y or yes to accept grid generated. Enter n or no to modify grid generated.
>Shows grid generated in plenum on workstation.
>Requests for acceptance. Enter y or yes to accept grid generated. Enter n or no to modify grid generated.
>Requests for amount of overlap above and below flat plate. Enter number of grid lines.
>Program outputs grid.in, q.save, bleed.inp, and an input file for PEGASUS, and then terminates.

This completes a typical interactive session with AUTOMAT. As noted earlier, sessions can be interactive or non-interactive. Non-interactive sessions do not show plots of the grid system generated on the screen, but provide the same outputs as the interactive sessions (namely, grid.in, q.save, and bleed.inp needed by OVERFLOW, and an input file for PEGASUS to get the fort.2 file which is also needed by OVERFLOW). Typically, an interactive session on a workstation requires about five minutes. Non-interactive session require less time (in fact, only only a few seconds on the Cray).

At the conclusion of a successful session with AUTOMAT, a job can be submitted to the Cray computer to run PEGASUS and then OVERFLOW. A typical grid system generated by using AUTOMAT for the inlet-bleed problem shown in Fig. 1 is given in Figs. 2 and 3. Typical solutions obtained by using OVERFLOW and PEGASUS are given in Figs. 4 and 5.

Internal Knowledge-Based System

This section outlines how AUTOMAT uses the input information to generate the grid system needed to obtain solutions.

The domain size is based on the following inputs: bleed-hole pattern (single hole or rows of holes arranged in staggered fashion), bleed-hole diameter, bleed-hole inclination, number of rows, spacing between hole centers in streamwise direction, spacing between hole centers in spanwise direction, plate thickness, with or without incident shock, shock-wave-generator half angle, and inviscid-shock-impingement point on flat plate.

Some rules in computing domain size (namely, L and H in Fig. 1) are as follows:

- $AB \geq 4$ (boundary-layer thickness),
- $AB \approx (2/3)\ AC$,
- $EF \approx (1/3\ \text{to}\ 2/3)\ DF$,
- the leading and trailing holes are at least five diameters away from the inflow and outflow boundaries, respectively.

Before describing the rules used to determine grid spacings, it is emphasized that grid generation has the greatest influence on the turn-around time of CFD simulations. Thus, it must be automated as much as possible in order to reduce turn-around time. The ability to automate grid generation depends very much on the grid structure selected. For the inlet-bleed problem, a grid structure based on overlapping, multi-block structured grids has a number of attractive features that make complete automation of the grid generation process possible. This is because if grids can overlap, then the domain regardless of its geometric complexity can be partitioned and molded into sub-domains with simple geometries that make both grid generation and hence its automation easy. Also, because each grid of the overlapping grid is optimized for a particular region, each of those grids is of high quality in terms of resolution, smoothness, and orthogonality. However, overlapping grids are not without problems. Many users have reported difficulties with conservation errors because the interpolation schemes used to transfer data between overlapping grids are not conservative. Others have observed errors associated with increased artificial diffusion in the overlapped region due to sudden changes in grid alignment and spacing. Thus, care must be exercised while generating such grids in order to minimize these errors.

In AUTOMAT, the grid structure used is the overlapping grid (see Figs. 2 and 3). Basically, an H-H grid structure is used for the region above the flat plate and in the plenum. In the bleed hole, a combination of O-H and H-H structures are used, and they overlap each other. The grids in the bleed holes also overlap the H-H grids above the flat plate and in the plenum. The reason for using the H-H structure above the plate and in the plenum is obvious. For the bleed hole, the reason for using the O-H structure is to capture the geometry of the hole boundary, and the reason for using the H-H structure at the hole center is to eliminate the centerline singularity associated with O-H grids.

The grid spacings in AUTOMAT are based on the following inputs: bleed-hole pattern (single hole or rows of holes arranged in staggered fashion), bleed-hole diameter, bleed-hole inclination, number of rows, spacing between hole centers in streamwise direction, spacing between hole centers in spanwise direction, with or without incident shock, shock-wave-generator half angle, and inviscid-shock-impingement point on flat

735

plate, boundary-layer momentum thickness, y^+ of first few grid point above flat plate, amount of cluster about incident and reflected shock, and amount of protrusion of grid in bleed hole above and below plate.

Some rules used to determine grid spacings are as follows (see Figs. 1 to 3):

- If results are to be compared for several different bleed-hole configurations, then the grid above the plate must be the same for all bleed-hole configuration. This is necessary in order to ensure that the flow above the plate is the same in the absence of bleed. Thus, the case with the largest bleed zone (i.e., from leading edge of first hole to trailing edge of last hole) will be used to generate grid above the flat plate.

- Above the flat plate, the grid system has an H-H structure.

- In a region about the bleed holes (i.e., 2D before and after the first and last hole), the grid spacing in the streamwise direction is uniform, at least for all practical purposes because of the following constraint: grid lines pass through the leading edge, center, and trailing edge of each bleed hole along the two symmetry boundaries. This constraint was imposed in order to facilitate plotting of the surface pressure along a line in the streamwise direction that passes through the centers of the bleed holes.

- Away from the bleed zone, the grid spacings in the streamwise direction first become coarser and coaser, and then nearly uniform as the inflow and outflow boundaries are approached.

- In the direction normal to the flat plate, grid points are clustered next to the wall to resolve the boundary layer. The first five grid points away from the wall are set so that y^+ is equal to 1, 2, 3, 4, and 5.

- Grid points are also clustered about the incident and the reflected shock waves in order to maintain their "sharpness" before and after interacting with the boundary layer.

- The grid in the plenum is also an H-H grid with grid points clustered near all walls and the bleed holes.

- The grid in the bleed hole is made up of two overlapping grids -- an O-H grid to capture the geometry of the hole boundary, and an H-H grid at its center to eliminate the centerline singularity associated with O-H grids.

- Grid spacings for the bleed-hole grids in directions normal to the plate match those above and below the flat plate in the region where they protrude above and below the plate.

- Grid spacings for the O-H grid in the bleed hole are clustered next the wall of the bleed hole.

- The aspect ratio of the grids is near unity for the grid in the bleed hole, especially in the region near the top of the flat plate.

- When α is not 90 degrees, grid spacing in the azimuthal direction is based on identical arc length, not degrees.

Additional details about the clustering as well as other aspects of the automated grid generation process can be obtained by perusing the Fortan program.

RESULTS

To illustrate the capabilities of AUTOMAT, the following steps were taken. First, AUTOMAT was used to generate several grid systems to demonstrate that geometric parameters such as bleed-hole inclination and the number of bleed holes can indeed be varied in parametric studies of the inlet-bleed problem. Next,

736

solutions were obtained to show that AUTOMAT does indeed provide all inputs files needed to do CFD simulations of the inlet-bleed problem and that the grid system generated can give meaningful solutions.

Figures 2 and 3 show the multi-block grid system generated by AUTOMAT for an inlet-bleed problem with four rows of normal bleed holes arranged in a staggered fashion. To further illustrate the automatic grid generation capabilities of AUTOMAT, grid systems were generated for the following two inlet-bleed problems without an incident shock wave impinging on the boundary layer: eight rows of normal bleed holes (Fig. 6), and eight rows of slanted bleed holes (Fig. 7). From these two figures, it can be seen that similar to the grid system shown in Figs. 2 and 3, grid lines are boundary conforming and are clustered near all solid walls whether the bleed holes are normal or slanted. Also, there are a sufficient number of grid lines clustered ahead of the first row of bleed holes and after the last row of bleed holes.

Figures 4 and 5 show the solutions obtained from PEGASUS and OVERFLOW by using the grid system (shown in Figs. 2 and 3) and other input files generated by AUTOMAT for an inlet-bleed problem with four rows of normal bleed holes arranged in a staggered fashion. These solutions have been shown to be grid independent as far as surface pressure and bleed rates into the bleed holes are concerned (ref. 16).

To further illustrate the parameters -- both geometric and operating -- that can be investigated, solutions were obtained for an inlet-bleed problem in which the bleed holes can be normal or slanted and the number of bleed holes can be one or three in tandem for a range of pressure ratios (P_b/P_∞). For the cases with a single bleed hole, the incident oblique shock wave was adjusted so that it would impinge at the center of the bleed hole under inviscid conditions. For the cases with three-holes-in-tandem, that shock wave was adjusted to impinge at the center of the middle hole. The grid systems used in this parametric study are not shown, but can be found in ref. 15. Some of the results obtained for this parametric study are shown, and are given in Figs. 8 to 13. Figures 8 and 9 show the effects of bleed-hole inclination on the surface pressure as a function of P_b/P_∞. Figures 10 and 11 show the effects of bleed-hole inclination on the surface pressure when there are three holes in tandem. Figures 8 to 11 can be compared with each other to examine the effects of having an upstream and a downstream bleed hole on the bleed process. Figures 12 and 13 show the flow coefficient in each bleed hole as a function of P_b/P_s, where P_s is the average static pressure over the bleed hole when there is no bleed (P_s has a different value over a different bleed hole due to the incident and the reflected shock waves). The flow coefficient is defined as the actual bleed rate divided by an ideal bleed rate. The ideal bleed rate assumes sonic flow in the entire bleed hole with stagnation pressure and temperature equal to those outside of the boundary layer. Analysis of the results shown in Figs. 8 to 13 can be found in ref. 15, and hence will not be repeated here.

The results presented in this section demonstrate that AUTOMAT can be used to automate CFD simulations of an inlet-bleed problem. These results also show that a relatively wide range of geometric and operating parameters of importance to inlet-bleed systems in supersonic aircraft can be investigated. The fast turn-around time afforded by AUTOMAT through the automatic generation of grid systems and other input files enables a broader range of design variables to be investigated.

SUMMARY

A CFD preprocessor, called AUTOMAT, was develop to generate all input files needed by OVERFLOW and PEGASUS to generate solutions for an inlet-bleed problem that possesses many of the most important geometric and operating parameters relevant to realistic inlet-bleed systems in supersonic aircraft. This paper described the inlet-bleed problem that AUTOMAT automatically sets up by along with the geometric and operating parameters that can be investigated. This paper also described the user interface as well as the knowledge-based system built into AUTOMAT to generate grid systems. Results obtained by using AUTOMAT, namely grid systems and computed solutions for the flowfield, demonstrated the usefulness of AUTOMAT.

ACKNOWLEDGEMENT

737

REFERENCES

1. Abrahamson, K.W. and Bower, D.L., "An Empirical Boundary Condition for Numerical Simulation of Porous Plate Bleed Flows," AIAA Paper 88-0306, January 1988.

2. Benhachmi, D. Greber, I., and Hingst, W., "Experimental and Numerical Investigation of an Oblique Shock Wave/Boundary Layer Interaction with Continuous Suction," AIAA Paper 89-0357, January 1989.

3. Chyu, W.J., Howe, G.W., and Shih, T.I-P., "Bleed Boundary Conditions for Numerically Simulated Mixed-Compression Supersonic Inlet Flows," AIAA Journal of Propulsion and Power, Vol. 8, 1992, pp. 862-868.

4. Paynter, G.C., Treiber, D.A., and Kneeling, W.D., "Modelling Supersonic Inlet Boundary Layer Bleed Roughness," AIAA Journal of Propulsion and Power, Vol. 9, No. 4, 1993, pp. 622-627.

5. Lee, J., Sloan, M.L., and Paynter, G.C., "Lag Model for Turbulent Boundary Layers over Rough Bleed Surfaces," AIAA Journal of Propulsion and Power, Vol. 10, No. 4, 1994, pp. 562-568.

6. Harloff, G.J. and Smith, G.E., "On Supersonic-Inlet Boundaru-Layer Bleed Flow," AIAA Paper 95-0038, January 1995.

7. Hamed A. and Lehnig, T., "An Investigation of Oblique Shock/Boundary Layer/Bleed Interaction," AIAA Paper 90-1928, July 1990.

8. Hamed, A. and Lehnig, T., "The Effect of Bleed Configuration on Shock/Boundary Layer Interactions," AIAA Paper 91-2014, June 1991.

9. Hamed, A., Shih, S.H., and Yeuan, J.J., "An Investigation of Shock/Turbulent Boundary Layer/Bleed Interactions," AIAA Paper 92-3085, July 1992.

10. Hamed, A., Shih, S.H., and Yeuan, J.J., "A Parametric Study of Bleed in Shock Boundary Layer Interactions," AIAA Paper 93-0294, Jan. 1993.

11. Hamed, A., Yeun, J.J., and Shih, S.H., "An Investigation of Shock Wave Turbulent Boundary Layer Interactions with Bleed through Slanted Slots," AIAA Paper 93-2992, July 1993.

12. Hahn, T.O., Shih, T.I-P., and Chyu, W.J., "Numerical Study of Shock-Wave/Boundary-Layer Interactions with Bleed," AIAA Journal, Vol. 31, No. 5, 1993, pp. 869-876.

13. Rimlinger, M.J., Shih, T.I-P., and Chyu, W.J., "Three-Dimensional Shock-Wave/Boundary-Layer Interactions with Bleed Through a Circular Hole," AIAA Paper 92-3084, July 1992.

14. Shih, T.I-P., Rimlinger, M.J., and Chyu, W.J., "Three-Dimensional Shock-Wave/Boundary-Layer Interaction with Bleed," AIAA Journal, Vol. 31, No. 10, 1993, pp. 1819-1826.

15. Chyu, W.J., Rimlinger, M.J., and Shih, T.I-P., "Effects of Bleed-Hole Geometry and Plenum Pressure on Three-Dimensional Shock-Wave/Boundary-Layer/Bleed Interactions," AIAA Paper 93-3259, July 1993.

16. Rimlinger, M.J., Shih, T.I-P., and Chyu, "Three-Dimensional Shock-Wave/Boundary-Layer Interactions with Bleed through Multiple Holes," AIAA Paper 94-0313, January 1994.

17. Benek, J.A., Buning, P.G., and Steger, J.L., "A 3-D Chimera Grid Embedding Technique," AIAA Paper 85-1523, 1985.

18. Benek, J.A., Donegan, T.L., and Suhs, N.E., "Extended Chimera Grid Embedding Scheme with Application to Viscous Flows," AIAA Paper 87-1126, 1987.

19. Buning, P.G. and Chan, W.M., "OVERFLOW/F3D User's Manual," NASA - Ames Research Center, 1991.

20. Baldwin, B. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, 1978.

21. Huang, P.G., Bradshaw, P., and Coakley, T.J., "Skin Friction and Velocity Profile Family for Compressible Turbulent Boundary Layers," AIAA Journal, Vol. 31, No. 9, 1993, pp. 1600-1604.

22. Steger, J.L. and Warming, R.F., "Flux-Vector Splitting of the Inviscid Gasdynamic Equations with Application to Finite-Difference Methods," Journal of Computational Physics, Vol. 40, No. 2, 1981, pp. 263-293.

23. Steger, J.L., Ying, S.X., and Schiff, L.B., "A Partially Flux-Split Algorithm for Numerical Simulation of Compressible Inviscid and Viscous Flow," Proceedings of the Workshop on Computational Fluid Dynamics, Institute of Nonlinear Sciences, University of California, Davis, California, 1986.

24. Walatka, P.P. and Buning, P.G., "PLOT3D User's Manual," NASA - Ames Research Center, 1989.

APPENDIX
DESCRIPTION OF INPUTS INTO AUTOMAT

As mentioned in the section on User Interface, for interactive sessions, the user will be prompted on the screen for the input data. For non-interactive sessions, the inputs are all contained in a namelist file, which can be modified by the user via any text editor (e.g., vi in unix). Also mentioned was that the namelist file required for non-interactive sessions is automatically generated after every interactive session to minimize effort needed to input data for future sessions. In this appendix, the input file, namelist.inp (default name given by AUTOMAT to the input file) is described in detail. A typical namelist.inp file is as follows:

```
no  ! Not an interactive session
$flow
  metric = .T. , fsmach =  1.60 , pinf =  61803.00 , tinf = 198.41,
  rgas = 287.200 , redl = 0.37130E+08 , momthick = 0.91400E-03 ,
  shock = .T. , thetwdge =  7.50 ,
  $end
$pattern
  axmaj =  0.2540E-02 , axmin =  0.2540E-02 , nhole =   4 ,
  pitchx =  0.8799E-02 , pitchy =  0.5080E-02 , halpha =  90.00 ,
  depth =  0.1016E-01 , shockx = 0.17598E-01 ,
  $end
$holepar
  finrad =  0.25 , fover =  3.00,
  dhmao =  0.5E-04 , dhmai =  0.1E-03 ,
  dhmio =  0.5E-04 , dhmii =  0.1E-03 ,
  delhent =  0.1495E-05 , delhmid =  0.2E-03 , delhexit =  0.5E-04 ,
  darc =  0.1814E-03 ,
  $end
$platepar
  finfx =  4.00 , foutx =  2.00 , ferri =  1.00 , ferro =  1.0 ,
  fskirtu =  2.00 , fskirtd =  2.00 , finfz =  6.00 ,
  din =  0.1270E-02 , dout =  0.7620E-03 , dbl =  0.6226E-03 ,
  delstream =  0.3E-03 , delspan =  0.1E-03 ,
  $end
$plenumpar
  flwall =  1.00 , finfz =  5.00 ,
  dlw =  0.8E-04 , dbw =  0.5E-04 ,
  dmidz =  0.1270E-02 , delhexit =  0.5E-04 ,
  $end
$overlapar
```

ntop = 4 , nbot = 10 ,
$end
$backpress
  prback = 0.3,
$end

An explanation of each parameter in the above namelist is given below.

- First line should always have the word "no". This alerts the program to use a non-interactive session.

- $flow: Parameters related to flow conditions.

  metric: T implies SI units, and F implies English units. All parameters having a numerical value should be given in the units chosen here.
  fsmach: freestream mach number at inflow (M_∞)
  pinf: pressure at inflow (P_∞)
  tinf: temperature at inflow (T_∞)
  rgas: gas constant for fluid
  redl: reynolds number per unit length
  momthick: momentum thickness of boundary layer (θ)
  shock: T or F indicating the presence of an incident shock wave
  thetwdge: half-angle in degrees of shock generating wedge (β)

- $pattern: Parameters related to bleed holes.

  axmaj: radius of hole on major axis (if bleed hole is inclined, then it appears as an ellipse on the flat plate; for normal holes, axmaj = axmin = D, D = diameter of hole)
  axmin: radius of hole on minor axis (same comment as that for axmaj)
  nhole: total number of holes (N)
  pitchx: spacing between hole centers in x (streamwise) direction (L_x)
  pitchy: spacing between hole centers in y (spanwise) direction (L_y)
  halpha: angle between bleed hole centerline and flat plate (α)
  depth: thickness of plate (L_t)
  shockx: location of where the incident shock wave impinges on the flat plate under inviscid conditions measured from the center of the first bleed hole (i.e., the origin of the coordinate system for the final grid)

- $holepar: Parameters used in generating the bleed hole grids

  finrad: the inner radius of the O-H bleed hole grid is given as $r_{imajor} = r_{omajor}*finrad = axmaj*finrad$
  fover: determines the overlap of the H-H and O-H grids in each bleed hole (fover specifies how many grid lines will be overlapped)
  dhmao: O-H grid spacing in radial direction at outer edge of major axis
  dhmai: O-H grid spacing in radial direction at inner edge of major axis
  dhmio: O-H grid spacing in radial direction at outer edge of minor axis
  dhmii: O-H grid spacing in radial direction at inner edge of minor axis
  delhent: spacing in z (depth) direction at the entrance of both hole grids
  delhmid: spacing in z (depth) direction at the midplane of both hole grids
  delhexit: spacing in z (depth) direction at the exit of both hole grids
  darc: O-H grid is equally-spaced in the circumferential direction (darc is the arclength of that spacing)

- $platepar: Parameters used in generating the flat plate grid

finfx: used to specify the x-location of the inflow

Procedure is as follows:
- Specify the height at which the shock wave enters the computational domain. In order to determine this height, choose it as some multiple of the boundary layer height at the inflow. $z_{shockinflow}$ = **finfx*blthick**
- Calculate the x position of the inflow plane based upon the equation of the straight line describing the impinging shock wave.

foutx: used to specify the x-location of outflow (same procedure are above; $z_{shockoutflow}$ = **foutx*blthick**)

Note: If there is no shock wave present, finfx and foutx represent the number of bleed hole diameters upstream of first bleed hole and downstream of last bleed hole to place the inflow and outflow planes; x_{inflow} = (finfx)*2*axmaj.

ferri, ferro: error contingencies for locating the inflow and outflow planes

In cases where the incident or reflected shocks are steep, the inflow and outflow boundaries might be located too close to the bleed holes. The locations are then forced to be as follows:
- x_{inflow} = (fskirt + **ferri**)*2*axmaj
- $x_{outflow}$ = (fskirt + **ferro**)*2*axmaj

skirtu, determines the size of the equi-spaced region upstream and downstream of the bleed hole
fskirtd: pattern; $x_{up-eqsui-start}$ = x_{hole1} - **fskirtu*2*axmaj**
finfz: determines the height of the domain; z_{top} = finfz*blthick (the code checks to ensure that the shock inflow position and the domain height are compatible. i.e. finfz > finfx)
din: spacing in streamwise direction at inflow plane
dout: spacing in streamwise direction at outflow plane
dbl: spacing at the top of the boundary layer (there is an equally-spaced region from z = blthick to 2*blthick)
delstream: spacing in streamwise direction above the bleed hole pattern
delspan: spacing in the spanwise direction.

• $plenmpar: Parameters controlling plenum grid generation

flwall: location of left wall in plenum; x = (flwall+fskirtu)*2*axmaj
finfz: location of lower wall in plenum; z = -(depth + finfz*axmaj*2)
dlw: spacing at left wall
dbw: spacing at bottom wall
dmidz: spacing in z direction at location halfway between the exit of the hole and the bottom wall
delhexit: spacing in the z direction at the top wall (matches the spacing at the outflow of the hole)

• $overlapar: Parameters governing the overlapped regions above the plate and into the plenum of the hole grids.

ntop: number of grid lines to extend the hole grids above the plate
nbot: number of grid line to extend the hole grids in the plenum

• $backpress

prback: ratio of back pressure to freestream pressure (P_b/P_∞)
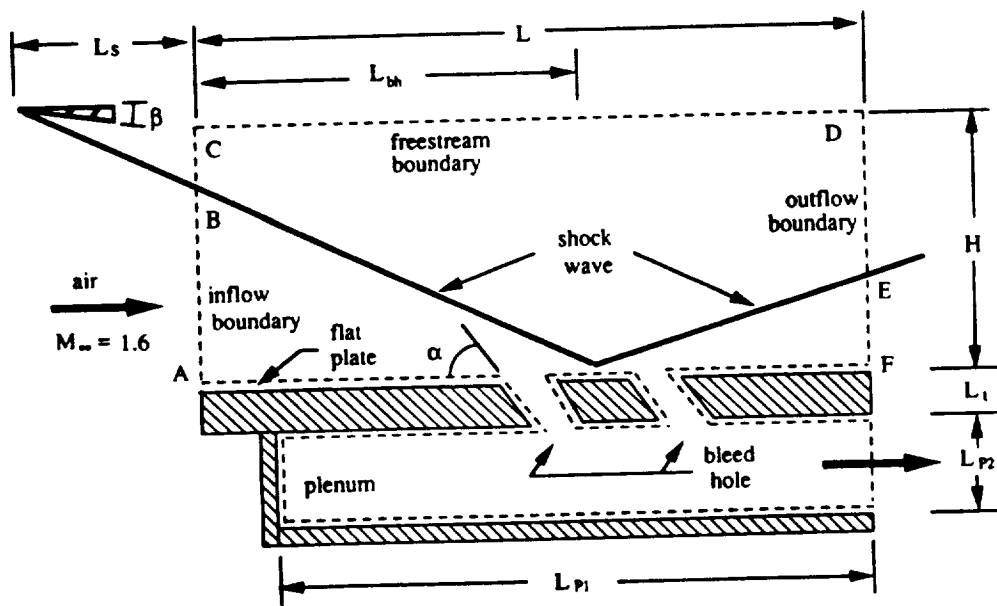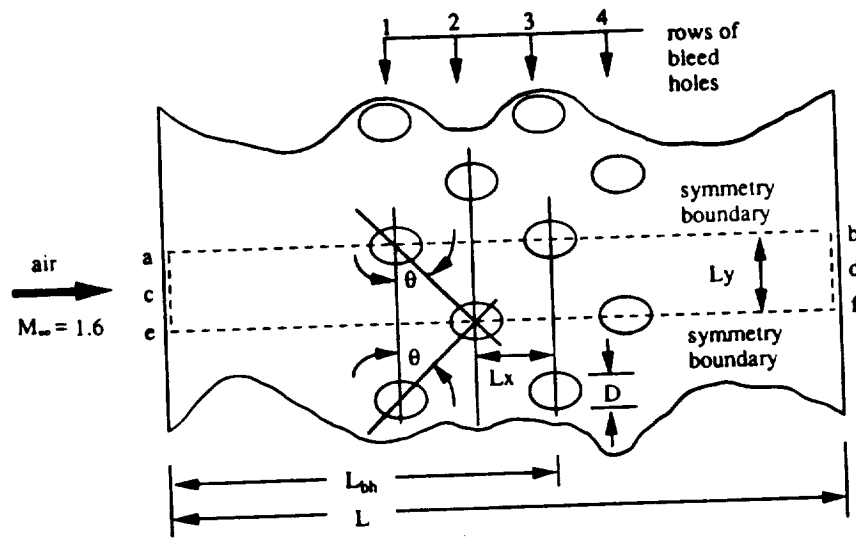
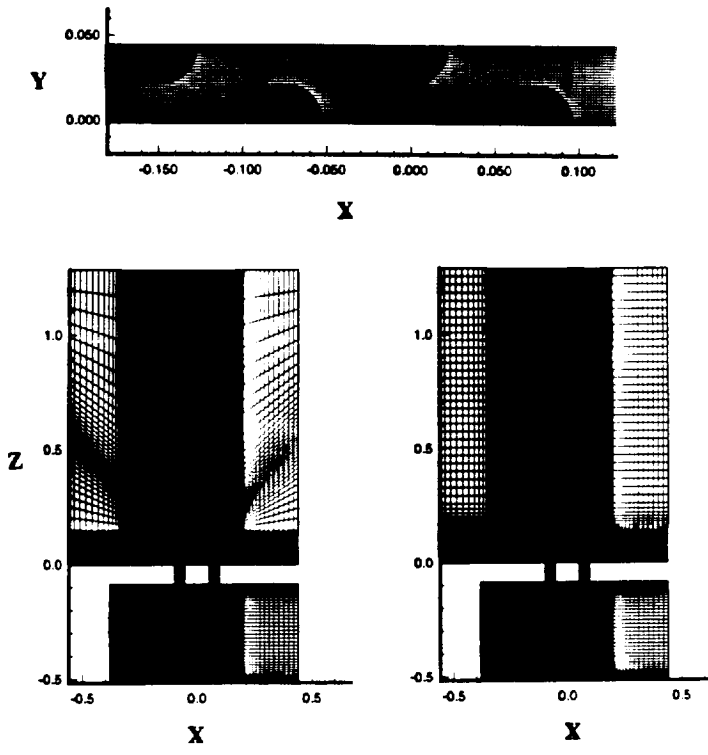Figure 1   Schematic of a typical inlet-bleed problem with four rows of bleed holes.

742
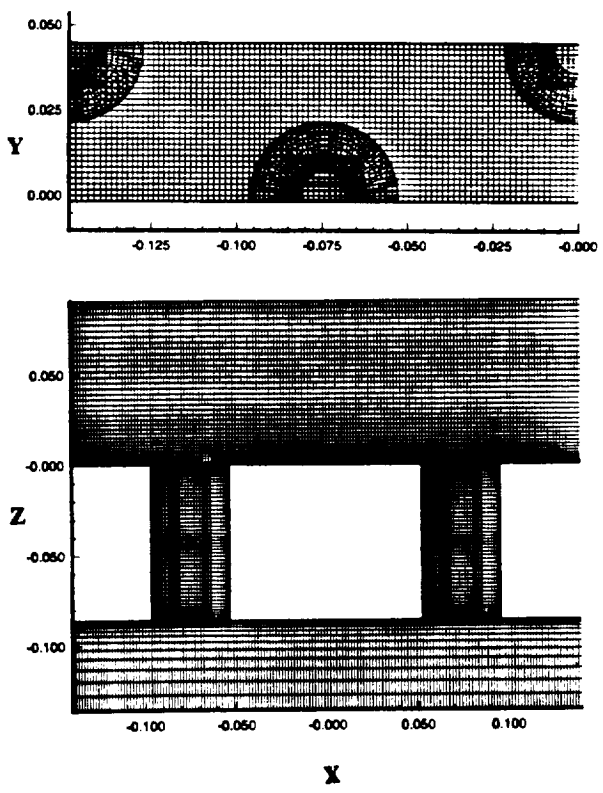
Figure 2   Typical grid system generated by AUTOMAT.

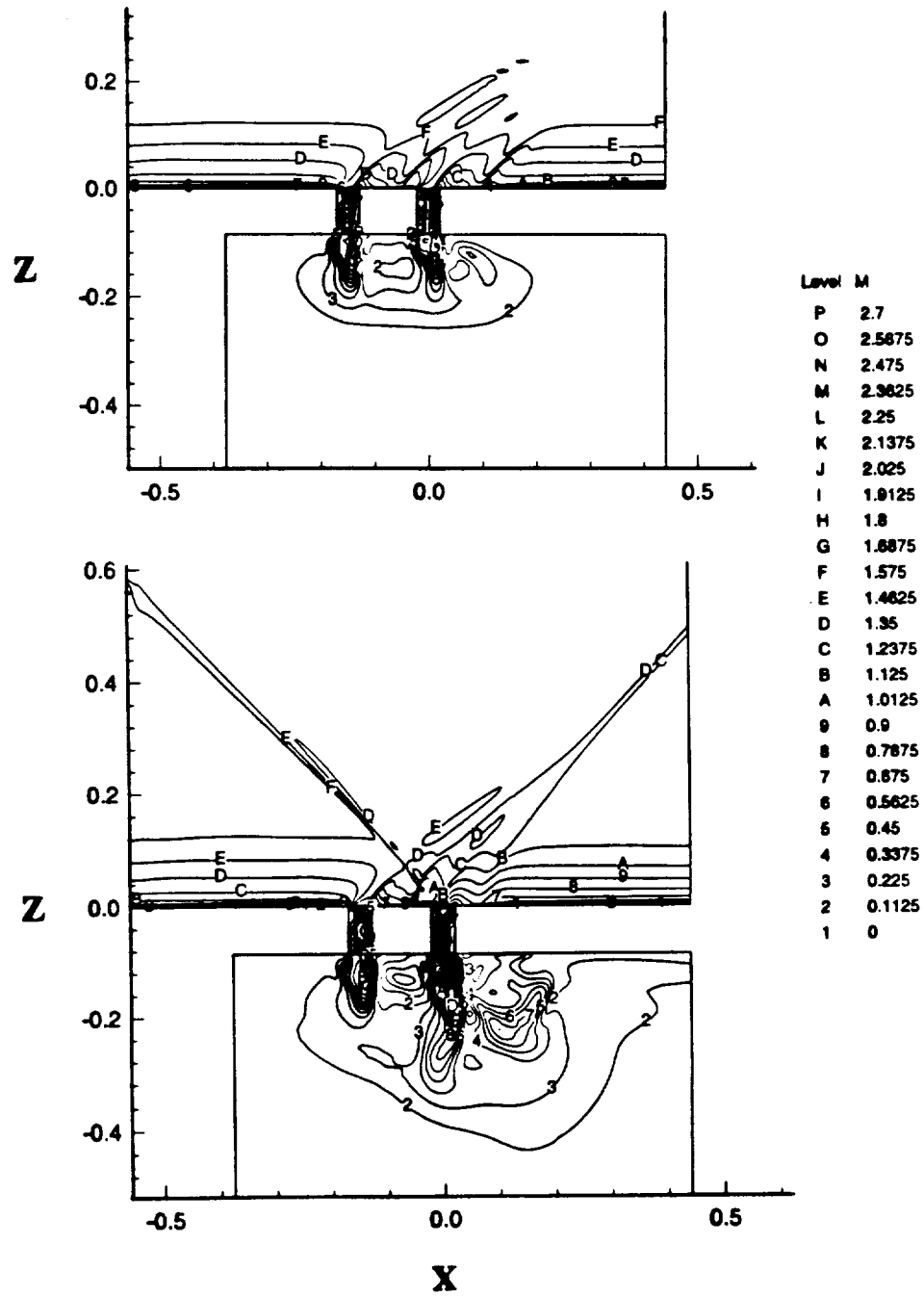Figure 3   Closeup view of grid system near a bleed hole.

Level	M
P	2.7
O	2.5875
N	2.475
M	2.3625
L	2.25
K	2.1375
J	2.025
I	1.9125
H	1.8
G	1.6875
F	1.575
E	1.4625
D	1.35
C	1.2375
B	1.125
A	1.0125
9	0.9
8	0.7875
7	0.675
6	0.5625
5	0.45
4	0.3375
3	0.225
2	0.1125
1	0

Figure 4   Typical solution (Mach number contours) generated by OVERFLOW with grid generated by AUTOMAT. Top figure: without incident shock. Bottom figure: with incident shock.

744
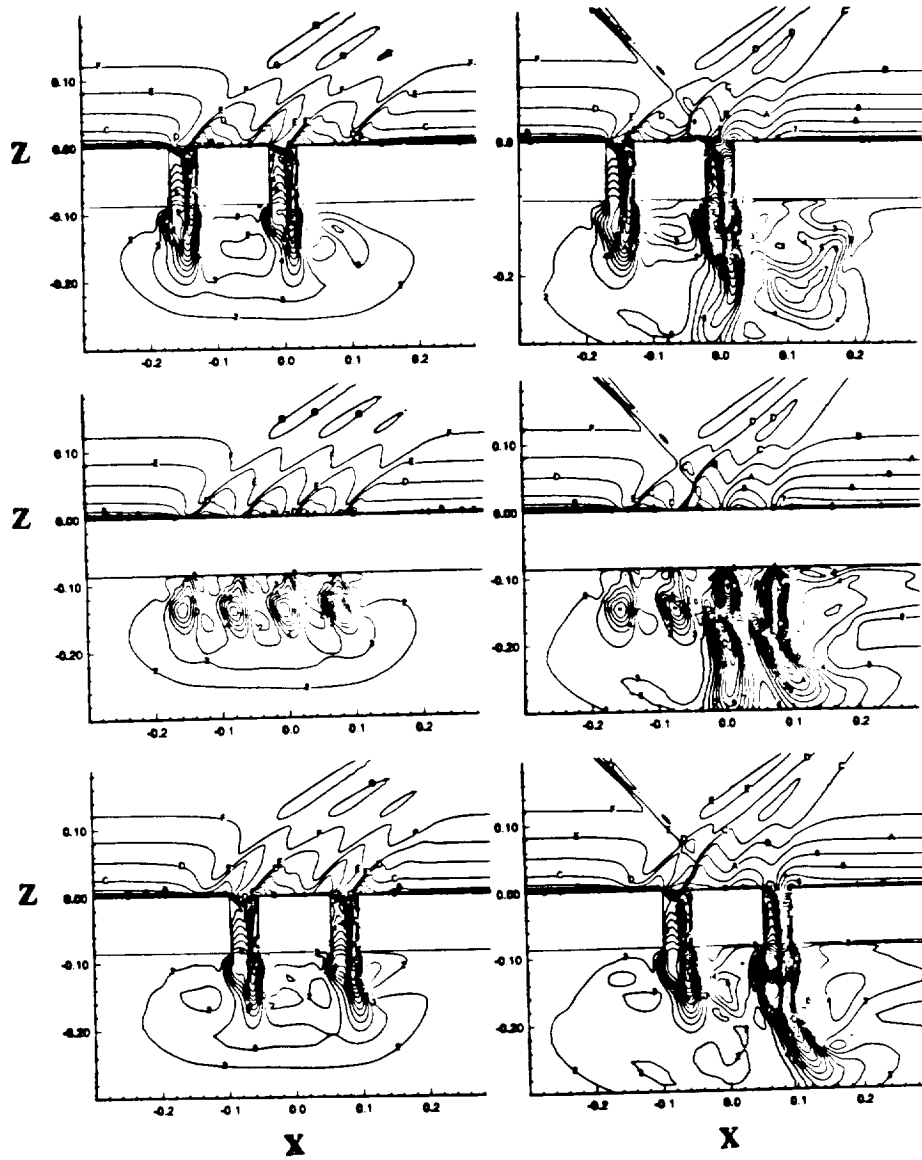
Figure 5   Typical solution (Mach number contours near a bleed hole) generated by OVERFLOW with grid generated by AUTOMAT. Left: without incident shock along three cross sections (top: a-b, middle: c-d, bottom: e-f). Right: with incident shock along three cross sections (top: a-b, middle: c-d, bottom: e-f). See Fig. 1 for definition of a-b, c-d, and e-f cross sections.

Grid-90" Normal Holes
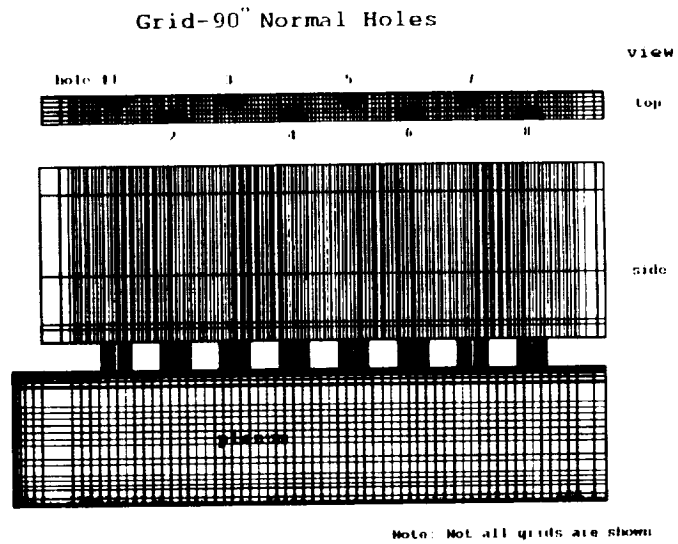
Figure 6   Grid system generated for an inlet-bleed problem with eight rows of normal bleed holes arranged in a staggered fashion (not all grid points shown).

Grid-30" Slanted Holes
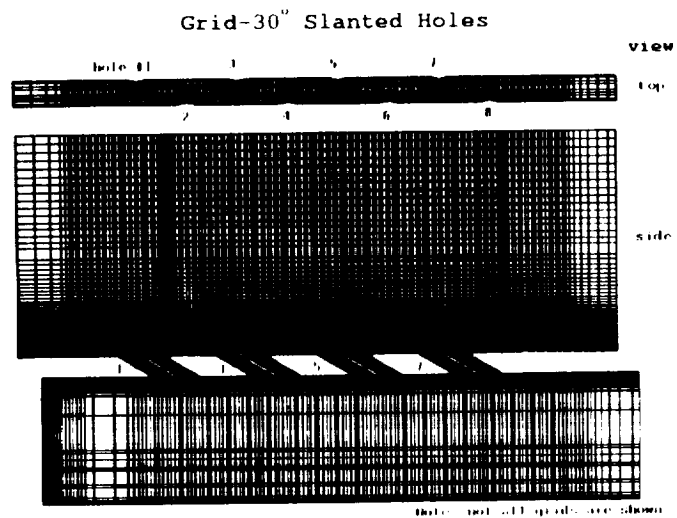
Figure 7   Grid system generated for an inlet-bleed problem with eight rows of slanted bleed holes arranged in a staggered fashion (not all grid points shown).
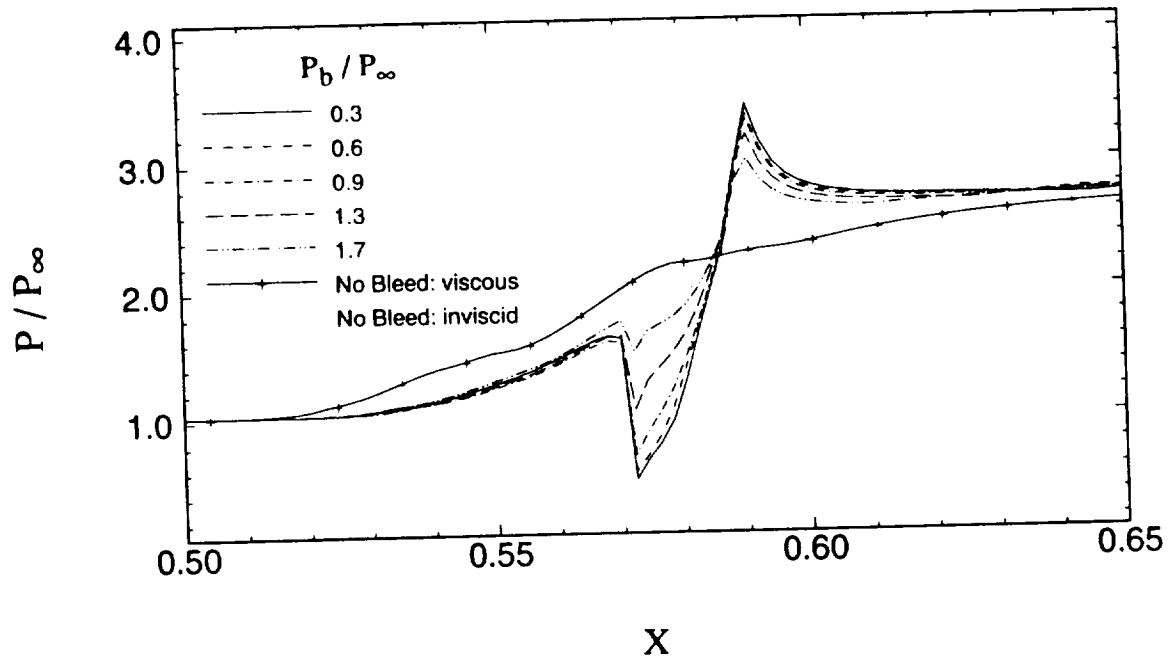
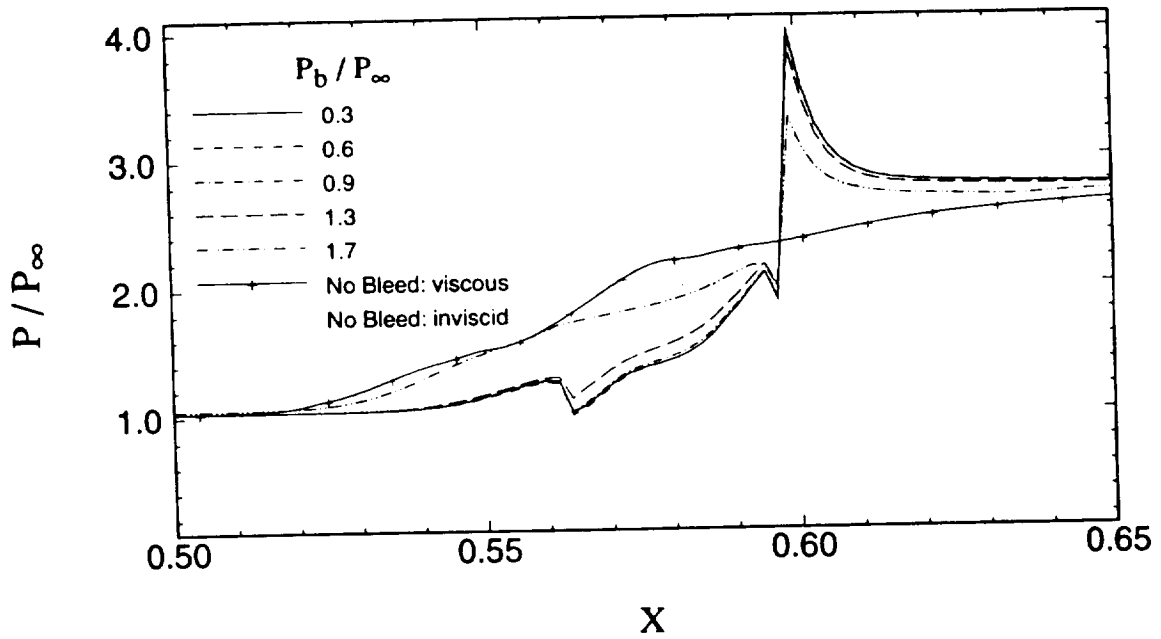Figure 8   Pressure along surface of flat plate from inflow to outflow passing through center of bleed hole for a single 90° bleed hole.

Figure 9   Pressure along surface of flat plate from inflow to outflow passing through center of bleed hole for a single 30° bleed hole.
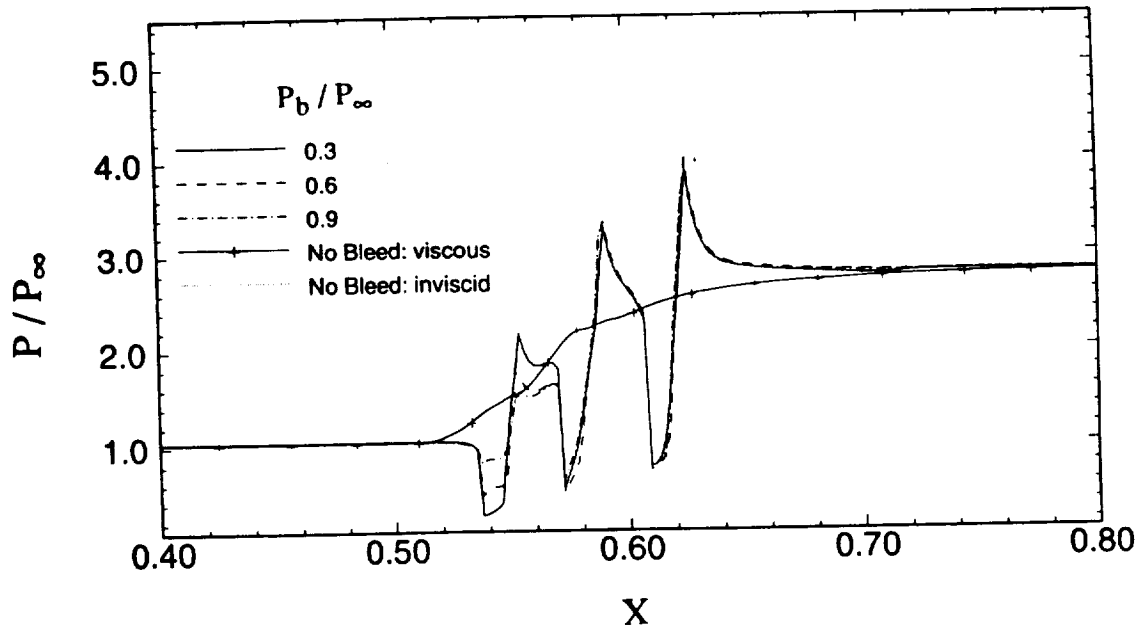
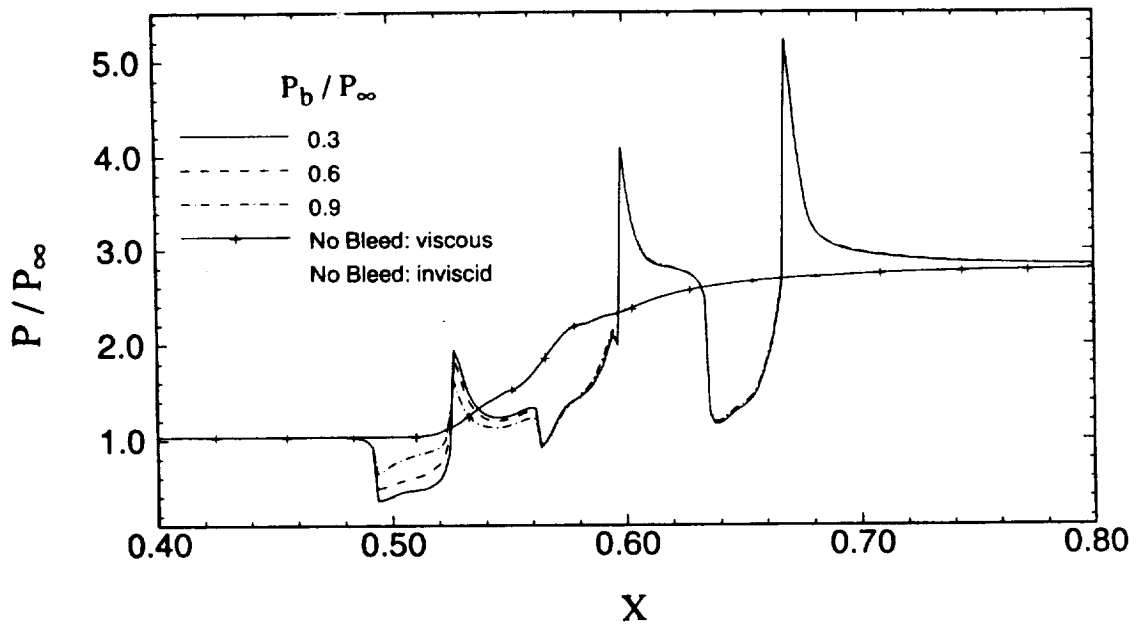Figure 10   Pressure along wall from inflow to outflow passing through center of bleed holes for three 90° holes.

Figure 11   Pressure along wall from inflow to outflow passing through center of bleed holes for three 30° holes.
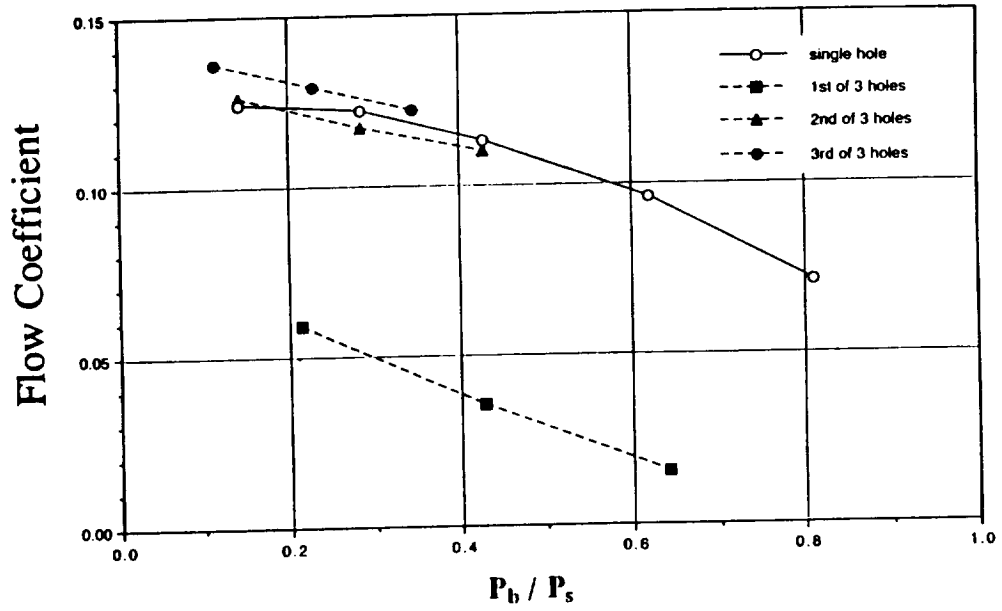
Figure 12   Flow coefficient as a function of P_b/P_s for 90° bleed hole or holes.
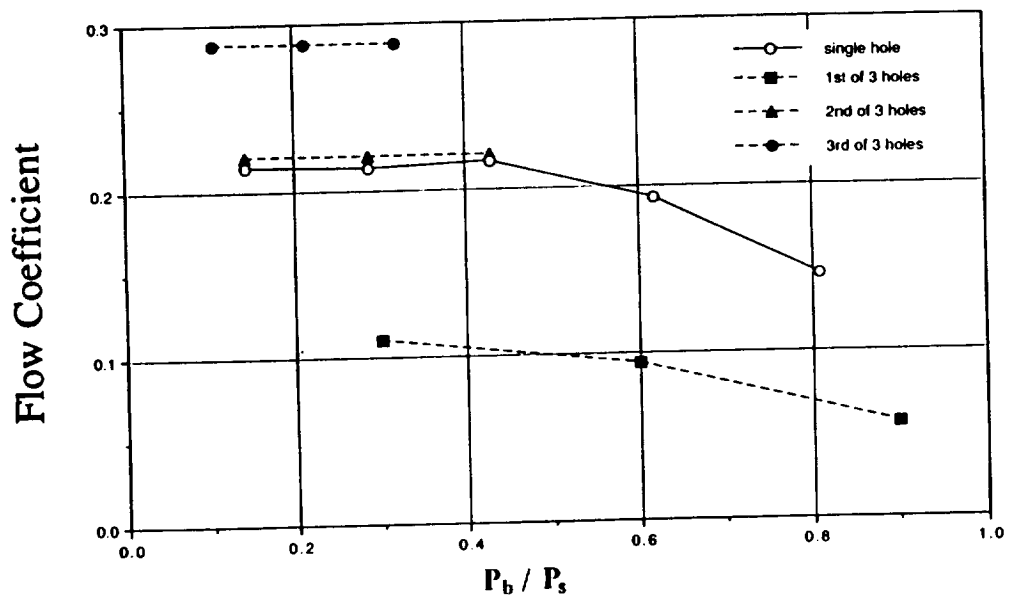

Figure 13   Flow coefficient as a function of P_b/P_s for 30° bleed hole or holes.