



Nearly Interactive Parabolized Navier-Stokes Solver for High Speed Forebody and Inlet Flows

*Thomas J. Benson, May-Fun Liou, William H. Jones, and Charles J. Trefny
Glenn Research Center, Cleveland, Ohio*

NASA STI Program . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) program plays a key part in helping NASA maintain this important role.

The NASA STI Program operates under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI program provides access to the NASA Aeronautics and Space Database and its public interface, the NASA Technical Reports Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.
- **CONFERENCE PUBLICATION.** Collected

papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services also include creating custom thesauri, building customized databases, organizing and publishing research results.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA STI Help Desk at 301-621-0134
- Telephone the NASA STI Help Desk at 301-621-0390
- Write to:
NASA Center for AeroSpace Information (CASI)
7115 Standard Drive
Hanover, MD 21076-1320



Nearly Interactive Parabolized Navier-Stokes Solver for High Speed Forebody and Inlet Flows

*Thomas J. Benson, May-Fun Liou, William H. Jones, and Charles J. Trefny
Glenn Research Center, Cleveland, Ohio*

Prepared for the
47th Aerospace Sciences Meeting
sponsored by the American Institute of Aeronautics and Astronautics
Orlando, Florida, January 5–8, 2009

National Aeronautics and
Space Administration

Glenn Research Center
Cleveland, Ohio 44135

This work was sponsored by the Fundamental Aeronautics Program
at the NASA Glenn Research Center.

Level of Review: This material has been technically reviewed by technical management.

Available from

NASA Center for Aerospace Information
7115 Standard Drive
Hanover, MD 21076-1320

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22161

Available electronically at <http://gltrs.grc.nasa.gov>

Nearly Interactive Parabolized Navier-Stokes Solver for High Speed Forebody and Inlet Flows

Thomas J. Benson, May-Fun Liou, William H. Jones, and Charles J. Trefny
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Abstract

A system of computer programs is being developed for the preliminary design of high speed inlets and forebodies. The system comprises four functions geometry definition, flow grid generation, flow solver, and graphics post-processor. The system runs on a dedicated personal computer using the Windows operating system and is controlled by graphical user interfaces written in MATLAB (The Mathworks, Inc.). The flow solver uses the Parabolized Navier-Stokes equations to compute millions of mesh points in several minutes. Sample two-dimensional and three-dimensional calculations are demonstrated in the paper.

Introduction

Optimization of hypersonic air-breathing vehicles requires consideration of the aerodynamic efficiency of a proposed configuration. This is especially true for the forward areas of the vehicle which condition and capture the propulsive stream, and generate aerodynamic forces. As a result, air inlet designs are driven to highly-integrated three-dimensional configurations. The inlet flow typically contains multiple shock waves and thick boundary layers on all surfaces. The interactions between the shocks and boundary layers cause traditional design techniques, such as the method of characteristics or streamline tracing, to be of limited utility for this class of inlets. Because the inlet must provide optimum performance over a wide speed range during acceleration, the inlet designer must consider a large matrix of geometric and flow variables. The large matrix of geometric and flow variables requires high-speed calculations for inlet design.

The overall objective of this study is to develop and link a series of nearly interactive computer programs for the preliminary design and aerodynamic evaluation of three-dimensional forebody and inlet configurations over a wide operating range of supersonic and hypersonic speeds. The system of programs should allow the designer to develop and evaluate a three-dimensional design in a matter of minutes. The final system will employ a commercially developed Computer Aided Design (CAD) package for geometry definition, optional programs for the flow grid generation, and may also include an optimizer program. This paper presents the current status of the system. At the center of this system of computer programs is a high-speed flow solver capable of accurately modeling three-dimensional shock-boundary layer interactions. The solver uses a single pass, spatial marching technique to solve the Parabolized Navier-Stokes (PNS) equations for supersonic and hypersonic flow through a specified geometry.

PNS solvers were developed in the 1970s and 80s to solve a variety of high-speed flow problems, References 1 to 3. A supersonic, three-dimensional, PNS analysis typically neglects the stream-wise diffusion terms of the full Reynolds-Averaged Navier-Stokes (RANS) equations to generate a solution algorithm that can be marched from one plane to the next on a structured grid. Because the solution is marched, and not globally relaxed as required by a RANS analysis, the solution technique is orders of magnitude faster than a comparable RANS analysis and requires orders of magnitude less computer storage.

The PNS solution technique does have its limitations, however. It can not model the flow past the terminal normal shock and it encounters serious stability problems in large regions of subsonic or

reversed flows, where neglected elliptical effects are important. A variety of solution techniques have been developed to model the flow in subsonic and reversed flow regions. For a supersonic or hypersonic inlet, large regions of subsonic or reversed flows are highly undesirable from a performance standpoint. It is normally not necessary for a hypersonic inlet preliminary design code to accurately model the details of a separation, but only to indicate that such conditions are present in the current design. The design must then be modified to eliminate the low speed flow to insure optimum inlet performance. A PNS analysis code is an ideal choice for a hypersonic inlet design tool; it is more accurate than a simple method of characteristics or stream tube analysis, yet orders of magnitude faster than a full RANS analysis.

The particular PNS analysis to be used in this design tool is the PEPSI-S computer program developed for NASA by Scientific Research Associates, Reference 4, in the early 1980s. PEPSI-S, an acronym for Parabolic Elliptic Streamwise Implicit-Supersonic, was chosen because it was originally developed for high-speed inlets and contains special logic to change boundary conditions from a flow boundary to a solid boundary in the marching direction as occurs at the centerbody, cowl, and sidewall leading edges. It also has special boundary conditions to model inlet bleed. This computer program has been verified for several important flow problems that occur in hypersonic inlets, including a cone at angle of attack, oblique shock boundary layer interactions, Reference 5, glancing shock boundary layer interactions, Reference 6, hypersonic compression corners, and a series of boundary layer bleed tests. It has also been verified against full high-speed inlet experiments, including the P-8 hypersonic inlet tested at NASA Ames, Reference 7, the Mach 5 inlet tested at NASA Glenn, Reference 8, and the Priced Option 2 inlet tested at CALSPAN as part of the National Aerospace Plane (NASP) project. The original flow solver was written in FORTRAN and consisted of nearly 12,000 lines of code. It was run on a variety of mainframe computers, including the Univac 1100, the IBM 360 and 370, and the Cray 1S and XMP. In the early 1990s, typical central processing unit (CPU) times on the Cray XMP for a problem with 1 million grid points was about 30 min. Studies have shown that the run time scales linearly with the number of grid points. The actual clock time to perform a calculation (turn-around time) was dependent on the number of other users in the queue for the mainframe. A turn-around time of 6 to 8 hr for a million point calculation was typical for the Cray XMP. Unfortunately, support for PEPSI-S was terminated in the early 1990s with the completion of the NASP project. Enhancements to the code to expand its geometrical capabilities and to include real gas effects had been coded but not extensively verified at the time of the termination.

Approach

The design system consists of four functions that run in rapid succession (1) geometry definition, (2) flow grid generation, (3) flow solution, and (4) graphical post-processing. Each function is performed by a separate computer program that is selected and launched by a graphical user interface (GUI) developed in MATLAB. The control GUI, shown in Figure 1, provides the user with some choices for geometry definition and grid generation. Each function program is controlled by its own GUI and passes information to other function programs through data files. The data files are archived to provide a record of past designs. The archived files can also be used as a starting point for a new design.

The preliminary design system is being developed on a stand-alone personal computer (2001 Dell Precision 330, with Pentium 4 chip) using the Windows operating system (OS). Windows OS was chosen for compatibility with the eventual CAD program for geometry definition. An older personal computer (PC) was selected for cost and availability considerations. The flow solver and grid generator were originally written in FORTRAN and compiled and executed under the Unix operating system for Cray computers (UNICOS). Some slight modifications to these programs were required to operate under MATLAB with Windows OS. The final design system will run on a much faster modern laptop computer running Windows.

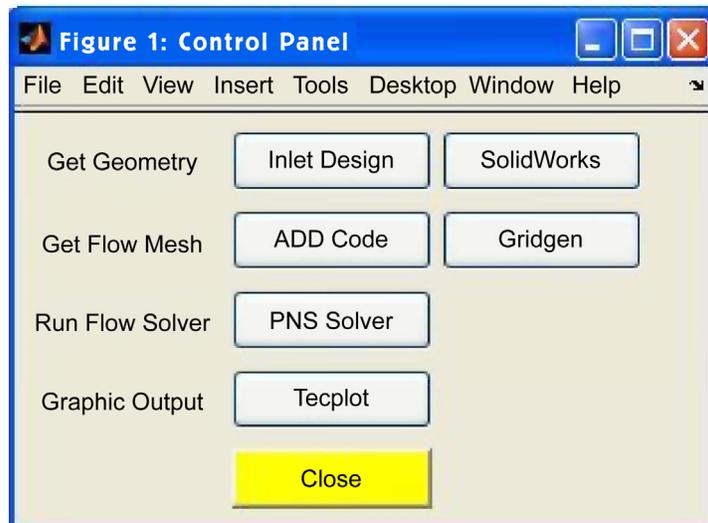


Figure 1.—Control graphical user interface (GUI).

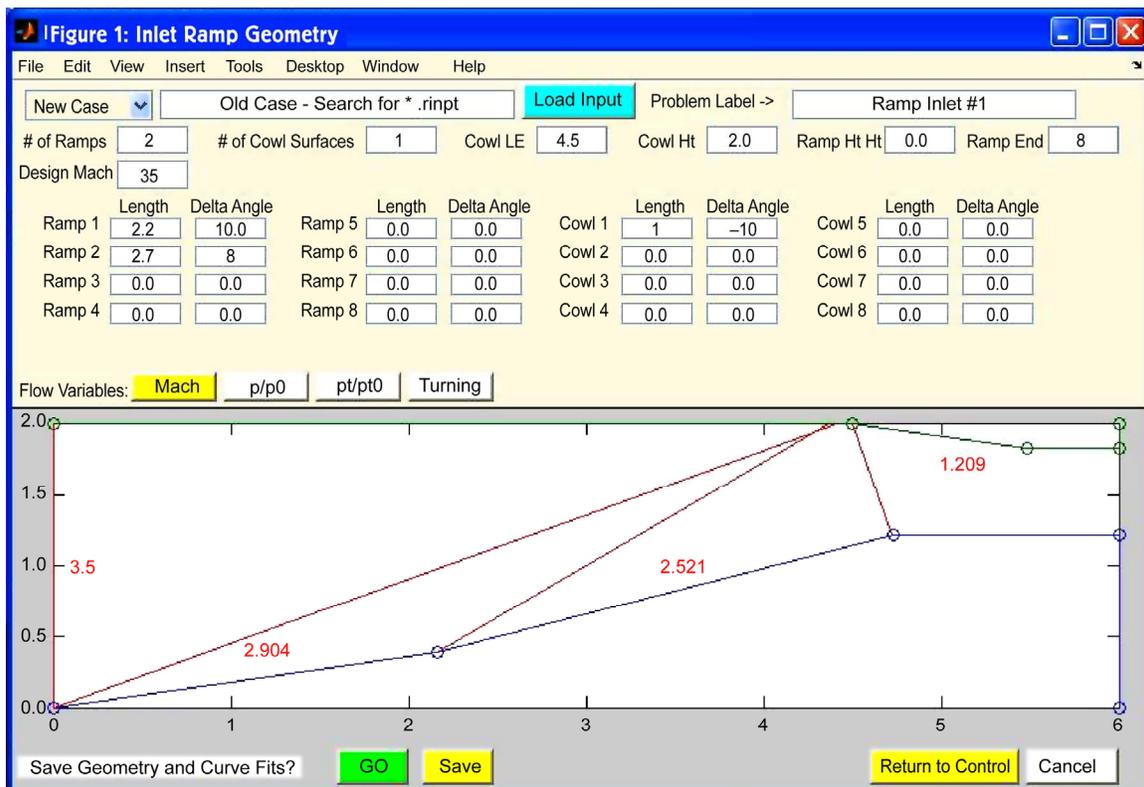


Figure 2.—Preliminary design program GUI.

The preliminary design process begins with the definition of the inlet geometry. A small interactive graphics program has been developed in MATLAB to help the user define the location of the inlet components. The user can vary the number, length, and angle of inlet ramps and cowl compression surfaces. The two-dimensional inviscid location of shock waves generated by the ramps and cowl are computed and displayed, as shown in Figure 2. Preliminary designs generated by the program can be saved for later modification and the program generates an input file for the grid generation program. More detailed geometry for a proposed configuration will eventually be developed using a commercially available CAD program such as SolidWorks (Geometric Software Solutions Co.).

The interactive geometry program is used to define the surface geometry of the inlet. A skeleton flow grid is then generated using a commercial grid generation program, such as Gridgen (Pointwise, Inc.), with the surface geometry as a boundary. The PNS flow solver interpolates on the skeleton grid to produce the actual computational grid. PEPSI-S has a requirement that the skeleton grid be orthogonal curvilinear. In previous uses of PEPSI-S, the Annular Diffuser Deck (ADD) code, Reference 9, was regularly used to generate skeleton grids. The ADD code is being retained as an alternate grid generation program for the design system. Input to the ADD code is generated by the interactive geometry program. The ADD code is executed by a MATLAB GUI and the output grid file created by the ADD code is stored using the same GUI.

With the successful generation of the skeleton flow grid, the user is ready to invoke the flow solver. Input to the flow solver is generated using another MATLAB GUI that is invoked from the control GUI of Figure 1. The flow solver GUI is shown in Figure 3. A GUI is used for the flow solver because of the increased speed offered by this method of input as compared to editing an input file. The input GUI is divided into five sections. The top section is used to locate previous input data files or restart files and to control the output from the PNS solver. The top section also includes “Status” and “Instruction” text boxes to inform the user of successful completion of the calculation, or to provide suggestions for error handling as required. The next lower section is used to specify the initial flow conditions to the solver. The middle section of the GUI provides a variety of input panels as selected by the user using various input buttons and pull-down menus. Figure 3 shows the boundary condition input panel. The section located below the middle section controls the grid resolution and grid data files. The bottom section is used to run the program, save results, and to call the output plotting program.

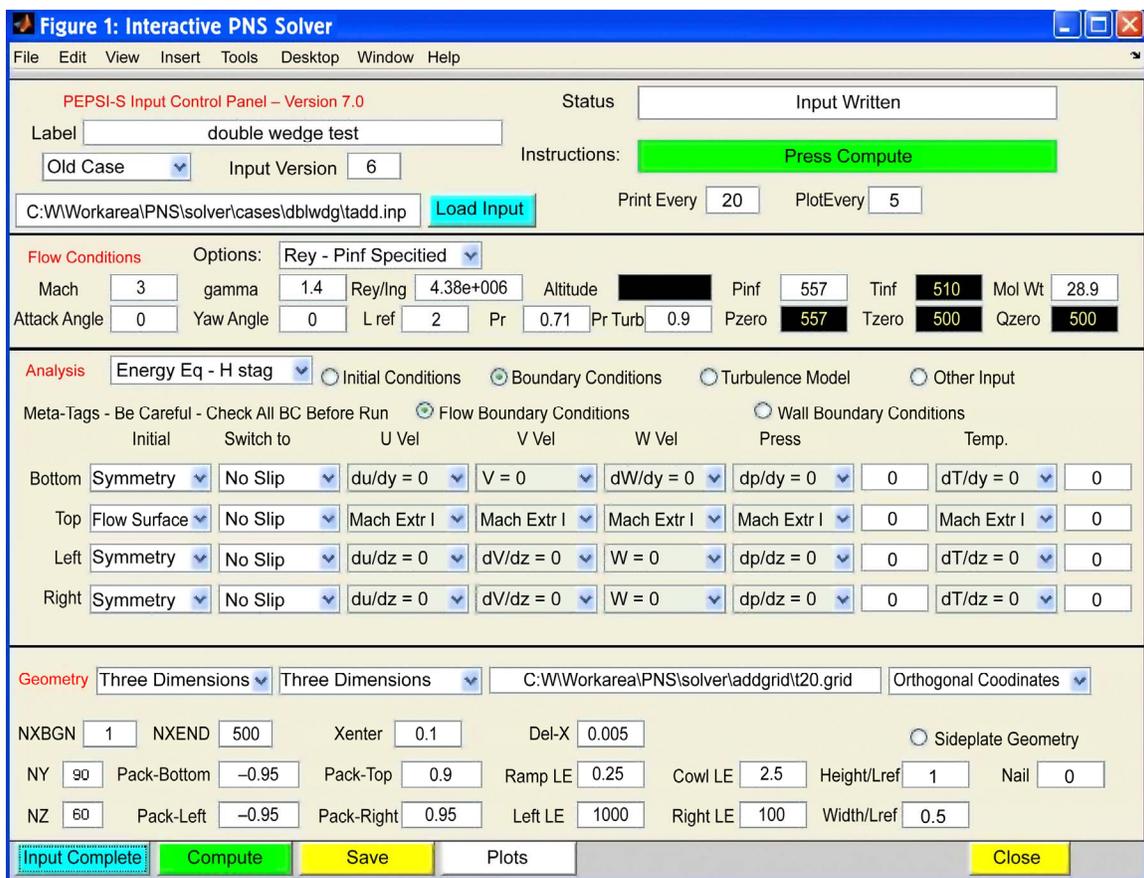


Figure 3.—PNS graphical user interface.

For a typical run, input for the flow solver is typed into the appropriate boxes. When all of the input variables have been specified, an input file is created by clicking on the “Input Complete” button at the lower left. The program is then executed by clicking on the green “Compute” button at the bottom. At the conclusion of the run, the middle section of the GUI displays average values of several flow variables at the last computational plane. If an error occurs during the calculation, it is noted in the “Status” box at the upper right so that the user can make modifications to the design. At the conclusion of a run, the user can choose to “Save” the input file for later retrieval using the GUI. The input file then provides a record of past designs and a starting point for a new design. The user can also save output from the flow solver as PLOT3D graphical files, as an ASCII output file, and as a binary restart file for the last plane calculated. The program can be restarted from any saved restart file as the entire flow field is initialized and the input GUI loaded with the conditions present at the designated plane. The graphical output is displayed using a commercial graphics package, such as TecPlot (TecPlot, Inc.), that is launched from the control GUI or from the flow solver GUI.

Sample Calculations

To demonstrate the power of the design system, let us follow the design of a rectangular inlet from preliminary sketch to full three-dimensional simulation. Using the interactive geometry program, a sample inlet is designed having two 7° ramps on the lower surface. The first ramp is 1.5 ft long and the second ramp is 4 ft long. The cowl is placed 2 ft above and 4 ft back from the ramp leading edge. A 3° internal compression ramp is added to the cowl. The preliminary geometry for this case was created and the input conditions for the ADD code were generated in about 2 min on the PC. The calculations were nearly instantaneous; the time was mostly spent entering data and doing visual checks of the results.

The geometry is updated and displayed for each entry and the final geometry is shown in Figure 4. The blue lines on the figure show the ramp side geometry, the green lines show the cowl geometry, and the red lines give the inviscid shock locations. The red numbers indicate the Mach number downstream of each shock wave. Note that this is a very bad inlet design, with multiple intersecting shock waves all falling inside the cowl. For this sample problem, the intent is not to design a practical inlet, but to exercise and time the design system, and to ensure that the flow solver can properly model complex flow features that could be present in a bad design. Were this a real design exercise, the front hinge of the second ramp and the cowl leading edge would be moved farther aft so that the ramp shocks do not intersect and pass outside the cowl.

A skeleton flow grid was generated for this inlet using the ADD code. The skeleton grid is 100 by 130 points and took 35 sec to generate on the PC. During the early 1990s, a similar size flow grid required about 30 min of CPU time on the Cray XMP. The grid is shown in Figure 5, with flow from left to right. The grid is packed on the upper and lower surfaces to provide increased grid resolution to the flow solver. Grid resolution is necessary to correctly model the boundary layers that form on the upper

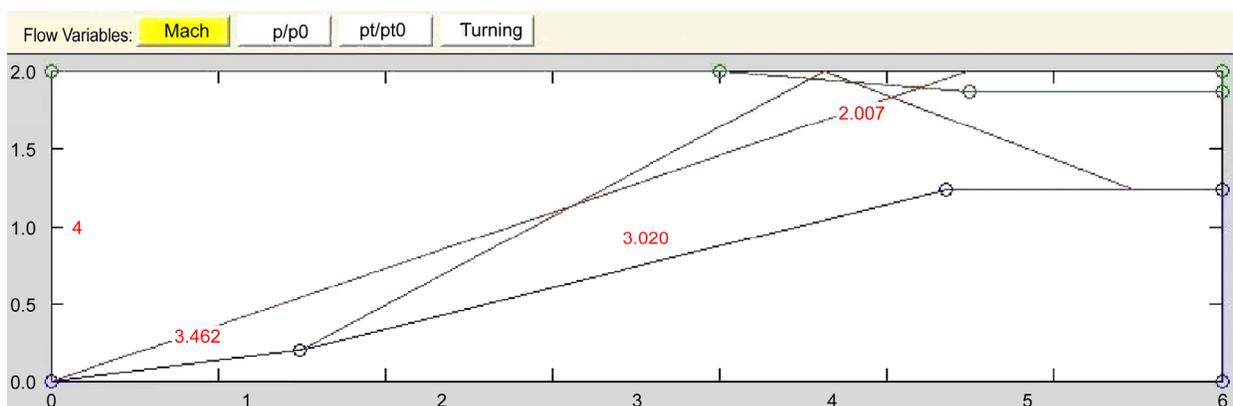


Figure 4.—Sample inlet geometry.

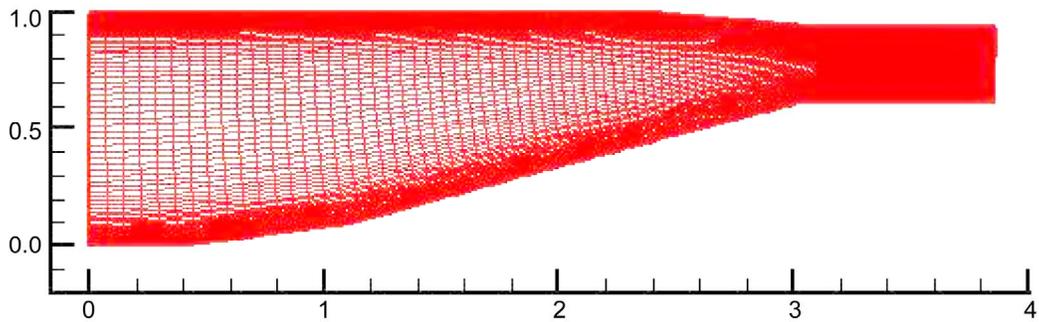


Figure 5.—Skeleton grid for inlet model.

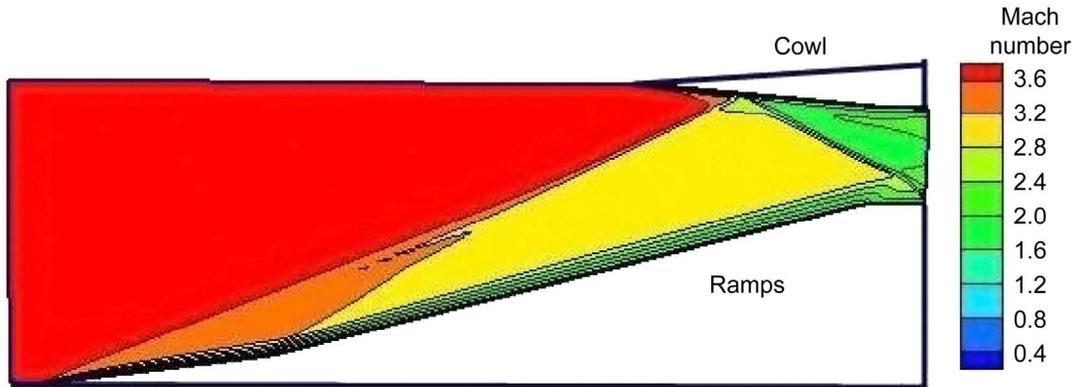


Figure 6.—Mach contours for the two-dimensional inlet model.

and lower surfaces. The numerical values on Figure 5 are different than Figure 4 because the flow grid is nondimensionalized by the cowl height. The skeleton grid file is normally stored in the same directory as the geometry file for a given case so that the grid file can be easily accessed by the flow solver and used for multiple flow calculations.

An initial, two-dimensional calculation of the flow through this inlet was performed at a free stream Mach number equal to 4.0 and a Reynolds number of 5 million per foot. The flow solver can be run with either Imperial units or metric units as selected by the user on the input GUI. A McDonald-Camarata mixing length turbulence model, Reference 10, was used for this sample calculation and the walls of the inlet were modeled as adiabatic surfaces. Grid resolution studies were performed to establish grid independence of the solution. The final calculation was performed on a 90 by 991 grid with a single restart file written at the midpoint of the marched solution. The total CPU time for this calculation was 15 sec. Approximately 2 min time was spent using the GUI on the first calculation to set up the problem by entering values for the free stream variables, specifying the boundary conditions, selecting the turbulence model, specifying the proper grid file, entering values for the grid resolution parameters, and saving the results. Subsequent calculations required less set up time because of the previously saved input files.

The Mach number contours through the inlet are shown in Figure 6, with red being Mach = 4 and blue being Mach = 0. The Mach number contours clearly indicate the growth of the boundary layer along the lower ramp surface and the upper cowl surface. The shock waves from the ramps and cowl are also indicated by the change in color contour. There are several complex compressible flow phenomena present in this sample case: intersection of the ramp shocks, intersection of the ramp shocks and cowl shock, reflection of the ramp shock from the cowl surface, and intersection of the reflected ramp shock with the expansion at the shoulder of the inlet at the right.

Some of these interactions are more clearly defined in the static pressure contours shown in Figure 7. In Figure 7, blue represents low pressure and red indicates high pressure. An abrupt change of color indicates a shock location. Because the static pressure is imposed through the boundary layer, a static pressure plot does not show boundary layer formation on the solid surfaces.

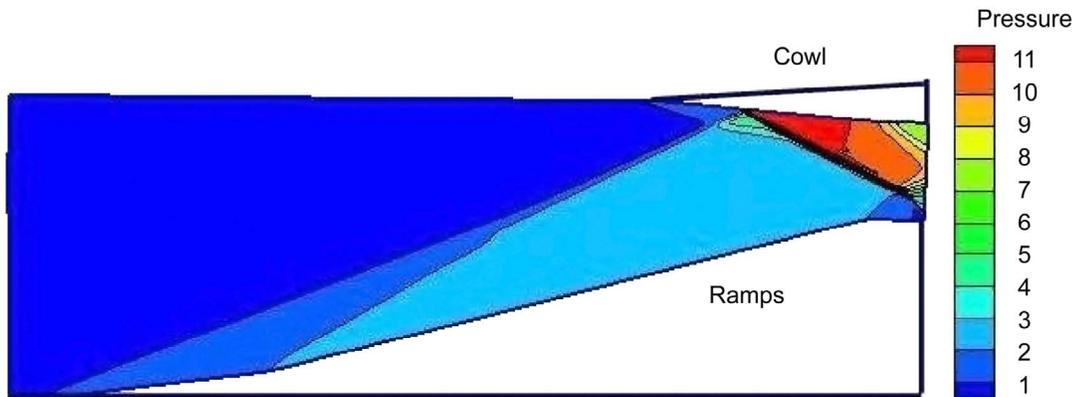


Figure 7.—Static pressure contours for the two-dimensional inlet model.

Following the two-dimensional calculations, the solver input file was modified to perform a three-dimensional design of the inlet. For the first three-dimensional design, a rectangular cross-sectional geometry was selected with the width equal to the cowl height. A solid sidewall was placed on each side, beginning at the ramp leading edge. The initial sidewall was a simple rectangular flat plate; later designs include a swept leading edge and a sidewall compression surface. A plane of symmetry exists along the center of the inlet, parallel to the sidewalls, so only one half of the flow domain needs to be modeled. Approximately 2 min were spent modifying the solver input file using the solver GUI to change from two-dimensions to three-dimensions and to specify boundary conditions and grid resolution on the sidewall surface. For this sample calculation, only the forebody ramp surfaces were calculated. A three-dimensional calculation was then performed on a 90 by 90 by 500 grid (4.05 million grid points) requiring 10 min on the PC. A calculation of this size would have required more than 2 hr of CPU time and a turn-around time of about a day on the Cray-XMP with the same solver during the 1990s. Some results of this calculation are shown in Figure 8.

Figure 8 shows Mach contours at four selected stream-wise planes. Mach = 4 is denoted by the red colors and Mach = 0 is blue. The flow enters from the lower left and moves towards the upper right. The compression ramps are at the bottom and the cowl surface would be at the top. The sidewall is located on the surface farthest from the viewer, while the symmetry plane is located on the near side. The contours indicate the shock wave generated by the first ramp by the change from red to orange, and the shock generated by the second ramp as the change from orange to yellow. Boundary layers along the ramp and sidewall surfaces are indicated by the thin predominately green bands. The bulge in the boundary layer along the sidewall is typically produced by the interaction of the sidewall boundary layer with the glancing shock from the ramp. Notice that this rather simple rectangular geometry produces a highly three-dimensional flow field because of the shock boundary layer interactions.

More details of the shock boundary layer interactions are given by the static pressure contours in Figure 9. Figure 9 shows static pressure contours at the same four stream-wise planes and same orientation as shown in Figure 8. Blue represents low pressure and red indicates high pressure. The change in color contours indicates the location of the shock waves. There is a very weak shock that is generated by the leading edge of the flat sidewall and this shock is seen to traverse the flow field from the sidewall to the centerline. The details of the glancing shock sidewall boundary layer interaction show that the pressure disturbance from the shock feeds upwards through the boundary layer ahead of the location of the free-stream inviscid shock. This behavior has been seen in verification cases of sidewall shock interactions, Reference 5.

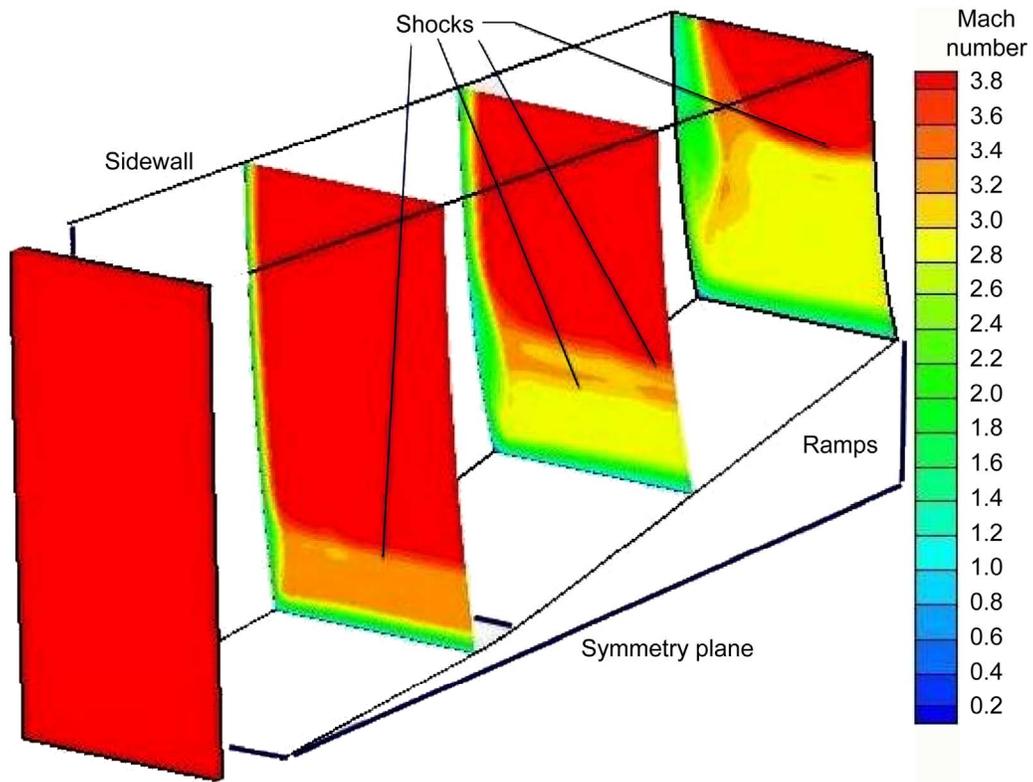


Figure 8.—Mach contours for the three-dimensional inlet model.

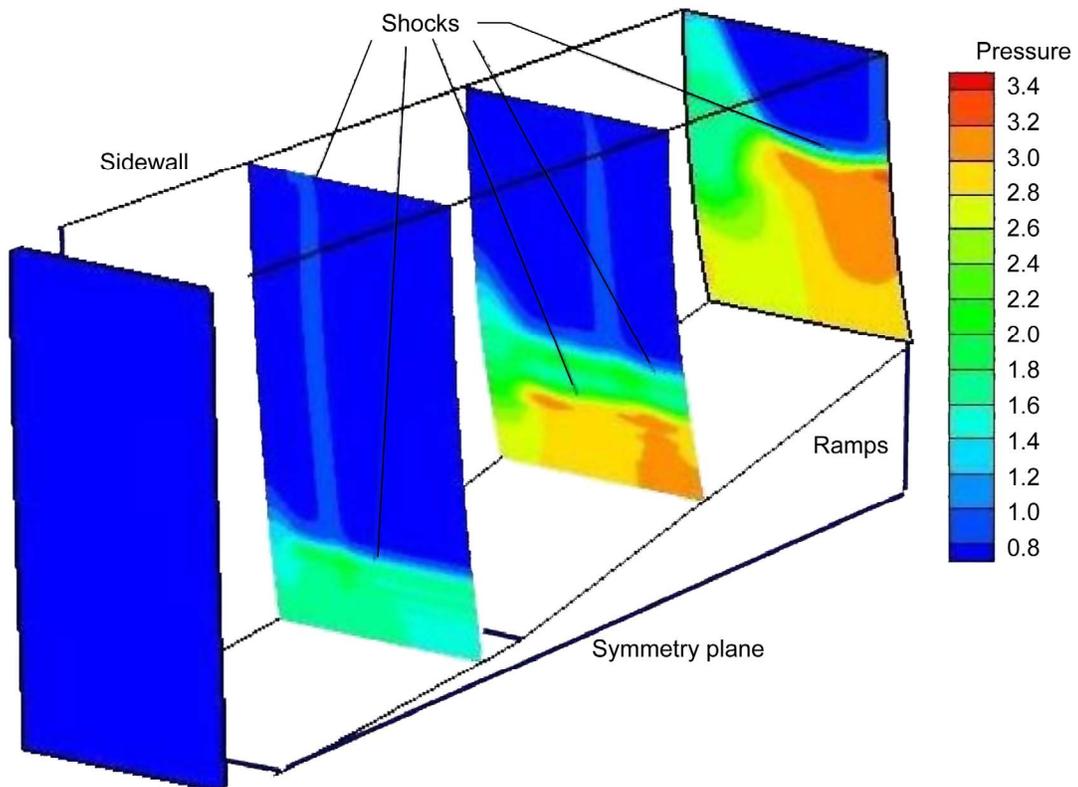


Figure 9.—Static pressure contours for the three-dimensional inlet model.

A final three-dimensional design case was initiated using the solver GUI and the solver input file from the previous case. For the new design, the flat plate sidewall was changed from a flat plate into a 7° sidewall compression ramp. This change required several minutes using the GUI to describe the geometry of the sidewall and to invoke a special option in the solver to handle the nonorthogonal grid geometry in the cross flow direction. A second 90 by 90 by 500 calculation was performed in 10 min time on the PC. Results of this calculation are shown in Figure 10.

In Figure 10, the orientation of the picture is the same as Figure 8; flow is from lower left to upper right, the plane of symmetry is nearer the viewer and the sidewall is away from the viewer. The compression wedge of the sidewall is indicated by the triangle near the top of the picture. Again, Mach = 4 corresponds to the color red and Mach = 0 corresponds to blue. Five planes of contours are shown in Figure 10. The free stream plane at the left indicates uniform flow. The plane just to the right of the free stream plane shows flow on the first ramp. Because the first ramp and the sidewall compression are both 7° , the flow field is symmetric about the far corner. There is a complex shock-on-shock interaction near the corner in addition to the shock boundary layer interactions on both the ramp and sidewall surfaces. At the far right, along the centerline of the inlet we see indications of flow separation (blue color) along the ramp surface. This separation is caused by the high adverse pressure gradient generated by the intersection of the sidewall compression shocks. In an actual application, the GUI would be used to change the design to eliminate this separation.

The speed of these calculations have allowed us to go from initial sketch to multiple three-dimensional designs in less than an hour. Such speed can also be used to analyze existing high-speed inlets. A test calculation was performed on the P-8 inlet, Reference 11, which was used extensively for code validation in the 1990s. The calculation started with geometric information from the report and proceeded through grid generation, flow solution using the solver GUI, and graphical analysis of the computation.

The free stream Mach number for this case was 7.4, the Reynolds number was 8.8 million per meter, the cowl height was 8.89 cm, forebody length was 82.3 cm, and the walls were held to a constant temperature of 283 Kelvin. The McDonald-Camarata mixing length turbulence model was used for this calculation. The computational grid was 90 by 80 by 1400 mesh (10.08 million grid points) and the calculation required about 30 min total computer time. Results from this calculation are shown in Figure 11.

Figure 11 shows Mach contours at four stream-wise planes and along the centerline of the inlet. The flow is from lower left to upper right and there is a large wedge forebody to the lower left which is not shown. The cowl is at the top and the lower cowl surface and the end of the forebody wedge curve inside the cowl. There is also a reverse swept sidewall on the near side that has been removed for viewing the flow field. Thick boundary layers have grown on the wedge surface and the cowl shock separates this boundary layer at the last computational plane at the far right in the corner formed by the sidewall and the wedge. This flow field compares favorably with previous calculations of this inlet, Reference 7, but was computed in 8 percent of the CPU time, and less than 1 percent of the turn-around time, required for the previous calculations.

The speed of the flow solver allows some other uses besides design and analysis. The flow solver supports two-equation turbulence models, so the system can be used in the development of improved boundary layer transition models for hypersonic application. The boundary conditions of the flow solver include mass removal and addition, so the system can be used to develop improved boundary layer bleed models, as long as elliptic effects are not important and can be neglected. The system can also be used to perform detailed computational studies of fundamental physics problems, such as the glancing shock boundary layer interaction problem.

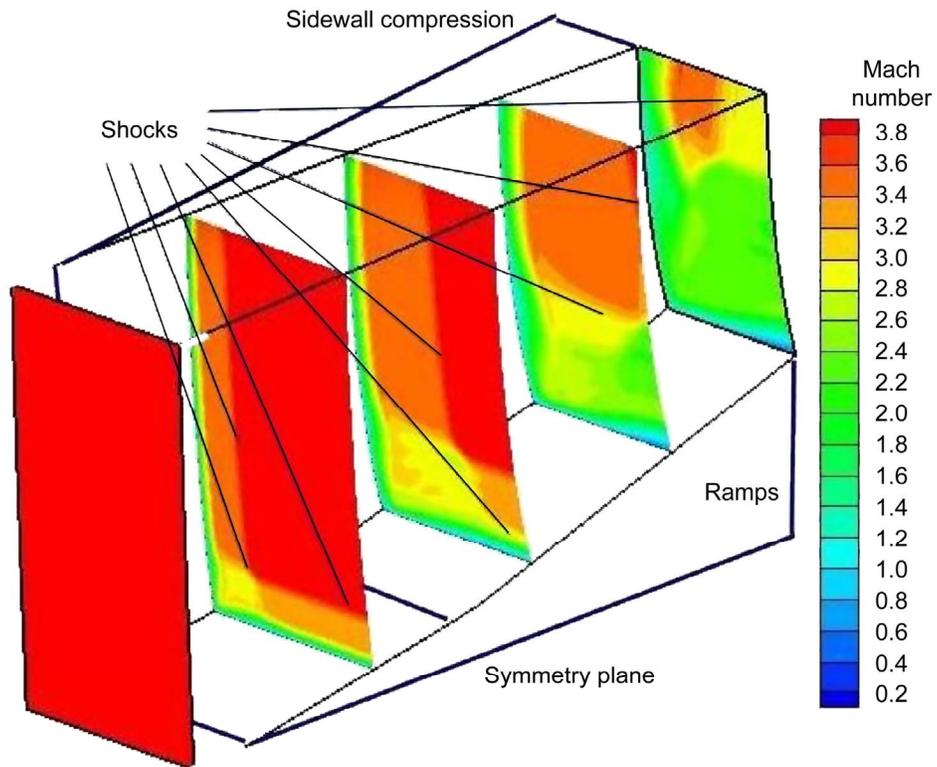


Figure 10.—Mach contours for the three-dimensional inlet model with sidewall compression.

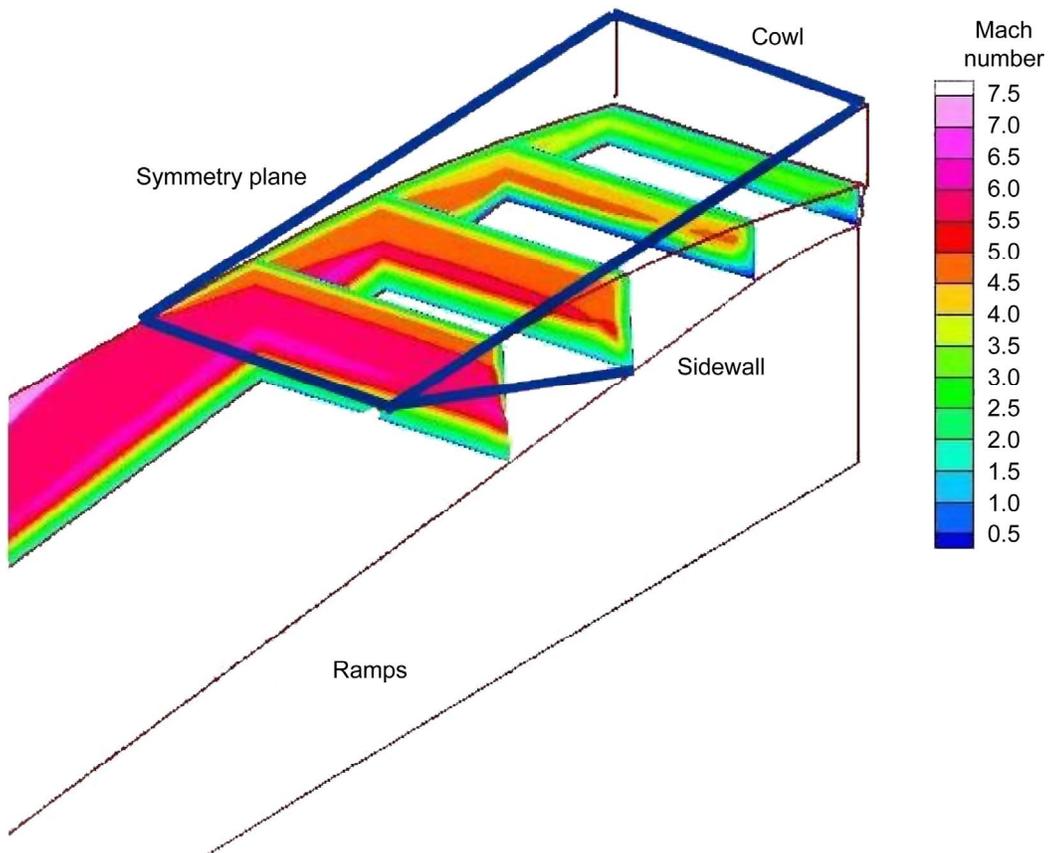


Figure 11.—Mach contours for the P-8 inlet.

Future Plans

The entire design system is currently under development and this paper has concentrated primarily on the development and use of the flow solver. Additional improvements are planned for the flow solver. For hypersonic applications, real gas effects associated with the excitation of the vibrational modes of nitrogen and oxygen molecules will be implemented and compared to experimental data. The solver was originally written for small core mainframe computers. To overcome the problems associated with a small core, the program writes and reads data from scratch files as the solution is marched downstream. The available storage on a modern PC far exceeds the storage available on the older mainframes. We intend to rewrite a portion of the solver to eliminate the data transfer. We expect a 20 to 30 percent speed increase from the solver with this improvement. We also intend to move the system to a more modern, high-speed laptop or dedicated PC with an associated speed increase expected. The goal for the flow solver is to be able to compute 1 million mesh points in 1 min; the current system can calculate 1 million mesh points in 2.5 min.

The current design system includes the interactive geometry program and has primarily employed the ADD grid generator. The final system will rely on CAD geometry development from SolidWorks and use commercial grid generation packages such as Gridgen. The flow solver has been modified to receive grid information from commercial packages. These features of the design system will have to be exercised and validated. Later versions of the system will also employ modern optimization techniques. The user can then choose to perform a design with the engineer in the loop or using the optimizer.

Conclusion

A new interactive preliminary design system is being developed to allow inlet designers to rapidly screen hypersonic forebody and inlet designs. The system employs a PNS solver to accurately model the three-dimensional shock boundary layer interactions that are present in this class of inlets. Modern, high-speed personal computers have sufficient computing power, speed, and storage to support such a system. On the current dedicated PC, one million grid points can be computed in 2.5 min. Because of the speed of the calculations, it is desirable to generate input to the program using a GUI written in MATLAB. The GUI passes information to the compiled FORTRAN modules of the solver and provides a system for storing and retrieving inputs from previous calculations. While the design system is currently under development, we have already begun to use the solver portion in an analysis mode to verify results, determine future requirements, establish timings, and build an accessible library of previous results.

References

1. Rudman, S. and Rubin, S.G., "Hypersonic Viscous Flow Over Slender Bodies With Sharp Leading Edges," *AIAA Journal*, vol. 6, no. 10, 1968.
2. Lubard, S.C. and Helliwell, W.S., "Calculation of the Flow on a Cone at High Angle of Attack," *AIAA Journal*, vol. 12, no. 7, 1974.
3. Schiff, L.B., and Steger, J.L., "Numerical Simulation of Steady Supersonic Flow," AIAA Paper 79-0130, 1979.
4. Buggeln, R.C., Kim, Y.N., and McDonald, H., "Computation of Multi-Dimensional Viscous Supersonic Flow," NASA CR-4021, 1986.
5. Benson, T.J., and Anderson, B.H., "A Study of Three-Dimensional Shock Wave/Turbulent Boundary Layer Interactions Present Within Aircraft Inlets," AIAA Paper No 84-1558, 1984.
6. Anderson, B.H., and Benson, T.J., "Numerical Solution to the Glancing Sidewall Oblique Shock Wave/Turbulent Boundary Layer Interaction in Three-Dimensions," AIAA Paper No 83-0136, 1983.
7. Benson, T.J., Bissinger, N.C., and Bradley, R.G., "Air Intakes for High Speed Vehicles," *Agard Advisory Report 270—Working Group 13*, Chapter 3, 1991.

8. Reddy, D.R., Benson, T.J., and Weir, L.J., "Comparison of 3D Viscous Flow Computations of Mach 5 Inlet with Experimental Data," NASA TM-102518, AIAA 28th Aerospace Sciences Meeting, 1990.
9. Anderson, O.L., "User's Manual for a Finite-Difference Calculation of Turbulent Swirling Compressible Flow in Axisymmetric Ducts With Struts and Slot Cooled Walls," USAAMRDL-TR-74-50, vol. I, 1974.
10. McDonald, H. and Camarata, F.J., "An Extended Mixing Length Approach for Computing the Turbulent Boundary Layer Development," *Proceedings of the Stanford Conference on Computation of Turbulent Boundary Layers*, vol. I, Stanford University, 1969, pp. 83–98.
11. Gnos, A.V., Watson, E.C., Seebaugh, W.R., Sanator, R.J., and Decarlo, J.P. "Investigation of Flow Fields Within Large-Scale Hypersonic Inlet Models," NASA TN D-7150, 1973.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

The 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 Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.
PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.

1. REPORT DATE (DD-MM-YYYY) 01-04-2009		2. REPORT TYPE Technical Memorandum		3. DATES COVERED (From - To)	
4. TITLE AND SUBTITLE Nearly Interactive Parabolized Navier-Stokes Solver for High Speed Forebody and Inlet Flows				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S) Benson, Thomas, J.; Liou, May-Fun; Jones, William, H.; Trefny, Charles, J.				5d. PROJECT NUMBER	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER WBS 599489.02.07.03.07.02.04	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration John H. Glenn Research Center at Lewis Field Cleveland, Ohio 44135-3191				8. PERFORMING ORGANIZATION REPORT NUMBER E-16889	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001				10. SPONSORING/MONITORS ACRONYM(S) NASA; AIAA	
				11. SPONSORING/MONITORING REPORT NUMBER NASA/TM-2009-215598; AIAA-2009-0711	
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Categories: 02, 07, 59, and 64 Available electronically at http://gltrs.grc.nasa.gov This publication is available from the NASA Center for AeroSpace Information, 301-621-0390					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT A system of computer programs is being developed for the preliminary design of high speed inlets and forebodies. The system comprises four functions: geometry definition, flow grid generation, flow solver, and graphics post-processor. The system runs on a dedicated personal computer using the Windows operating system and is controlled by graphical user interfaces written in MATLAB (The Mathworks, Inc.). The flow solver uses the Parabolized Navier-Stokes equations to compute millions of mesh points in several minutes. Sample two-dimensional and three-dimensional calculations are demonstrated in the paper.					
15. SUBJECT TERMS CFD (computational fluid dynamics); Supersonic inlet analysis; Hypersonic forebody analysis					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			19b. TELEPHONE NUMBER (include area code)
U	U	U	UU	18	STI Help Desk (email:help@sti.nasa.gov) 301-621-0390

