NOTICE

THIS DOCUMENT HAS BEEN REPRODUCED FROM MICROFICHE. ALTHOUGH IT IS RECOGNIZED THAT CERTAIN PORTIONS ARE ILLEGIBLE, IT IS BEING RELEASED IN THE INTEREST OF MAKING AVAILABLE AS MUCH INFORMATION AS POSSIBLE.
NUMERICAL SIMULATION OF SUPERSONIC INLETS USING A THREE-DIMENSIONAL VISCOS FLOW ANALYSIS

B. H. Anderson and C. E. Towne
Lewis Research Center
Cleveland, Ohio

Prepared for the Eighteenth Aerospace Sciences Meeting sponsored by the American Institute of Aeronautics and Astronautics Pasadena, California, January 14-16, 1980
NUMERICAL SIMULATION OF SUPERSONIC INLETS USING A THREE-DIMENSIONAL VISCOUS FLOW ANALYSIS

Bernhard If. Anderson* and Charles E. Towne**
NASA-Lewis Research Center
Cleveland, Ohio

Abstract

A three-dimensional, fully viscous computer analysis, which retains the viscous nature of the Navier-Stokes equations, was evaluated to determine its usefulness in the design of supersonic inlets. This procedure takes advantage of physical approximations to limit the high computer time and storage associated with complete Navier-Stokes solutions. Computed results are presented for a Mach 3.0 supersonic inlet with bleed and a Mach 7.4 hypersonic inlet. Good agreement was obtained between theory and data for both inlets. Results of a mesh sensitivity study are also shown.

Nomenclature

\( h_0 \) total enthalpy
\( h_1, h_2, h_3 \) metric coefficients
\( l_m \) mixing length
\( L_{REF} \) reference length (35.56 cm (14 in.) for M3 inlet, 18.33 cm (7.22 in.) for P8 inlet)
\( m_{BL} \) bleed mass flow
\( m_c \) capture mass flow
\( M \) free-stream Mach number
\( p \) static pressure
\( p_{REF} \) reference pressure (28.5 N/m² (58.8 psf) for M3 inlet, 701 N/m² (14.6 psf) for P8 inlet)
\( Re \) Reynolds number per unit length based on free-stream (reference) conditions
\( u_{REF} \) reference velocity (640.2 in/sec (2100 ft/sec) for M3 inlet, 1222 m/sec (4008 ft/sec) for P8 inlet)
\( u, v, w \) velocities in computational coordinates
\( x, y, z \) cartesian coordinates
\( \gamma \) ratio of specific heats
\( \delta_b \) boundary layer thickness
\( \kappa \) Von Kármán constant, 0.43
\( \mu \) effective viscosity, laminar + turbulent
\( \mu_T \) turbulent viscosity

*Head, Aerodynamics Analysis Section, Member, AIAA
**Aerospace Engineer, Member, AIAA

Introduction

The design of three-dimensional supersonic inlets is a difficult task in view of the wide operating range over which good performance is desired. Good agreement was obtained between theory and data for both inlets. Results of a mesh sensitivity study are also shown.

The foregoing considerations suggest that inlet design technology would benefit from a detailed and accurate flow field calculation procedure that includes shock-boundary layer interaction effects. Within rectangular inlets operating at supersonic speeds, viscous effects can be classified into three types: (1) incident shock reflections on the centerbody and cowl, (2) glancing shock interactions which take place along the inlet sidewalls, and (3) corner flow interactions. Calculation procedures which are either two-dimensional or do not account for boundary layer effects have limited application towards understanding these viscous interactions.

A number of experimental and analytical papers have been published dealing with these shock interactions. The analysis and comparison with data of an incident shock wave reflection within an axisymmetric inlet have been accomplished by Fukuda, Hingst and Reshotko and Hingst and Towne. In these analyses, control volume methods were used to solve for the properties downstream of the interaction zone given the conditions upstream of the interaction. A similar approach was used by Paynter for the analysis of the weak glancing sidewall interaction. These analyses yield correct trends and represent useful design techniques. An experimental study of the glancing sidewall interaction was done by Oskam, Vas and Bogdánszki and provides a detailed flow description from which to verify analyses. Except for the full Navier-Stokes analysis of the shock-corner boundary layer interaction by Hingst and McCormack most of the analyses...
of this phenomenon are inviscid and provide no information as to the viscous interactions. There are however, a number of experimental reports providing data on the viscous corner interaction which are helpful in understanding the flow structure itself.

Although full Navier-Stokes procedures would provide the necessary generality to predict the flow within three-dimensional supersonic inlets, the required computer time and storage would be prohibitive in terms of present computer technology. Combined viscous-inviscid interaction analyses of the type suggested by Keymer and Hickcox for axisymmetric inlets could be implemented for three-dimensional inlets; however, the coupling procedures would be very complex. An attractive alternative analysis would retain the general three-dimensional viscous nature of the Navier-Stokes equations but would take advantage of physical approximations to limit the high computer run time and storage associated with a complete Navier-Stokes solution. In the present analysis, the assumption is made that there is a primary flow direction and that diffusion in this direction can be neglected. In this manner a set of equations are produced for fully viscous, predominantly supersonic, three-dimensional flow which can be solved by an efficient forward marching technique.

This paper represents the first in a series of studies to evaluate the marching analysis of Huggins, Kroskovsky, and McDonald, designated PEPSIS, and to determine its usefulness in the design of supersonic inlets. Two inlets were chosen for this study; a Mach 3.0 configuration which obtains its compression by means of a crossed shock structure, and a Mach 7.4 hypersonic inlet. Efforts to date have concentrated primarily on the ramp/cowl shock wave boundary layer interactions.

Governing Equations

In this study, the inlet flow field is computed by a spatial marching method which solves a simplified form of the three-dimensional Navier-Stokes equations. A curvilinear orthogonal coordinate system is used with coordinate directions x, y, and z and corresponding metric coefficients $h_1$, $h_2$, and $h_3$. Here x is defined as the streamwise or marching direction, and y and z are the two cross-flow directions. The equations are first time-averaged so that they apply to both laminar and turbulent flow. Viscous terms are simplified by using an order-of-magnitude analysis. In particular, in order to allow the use of a marching procedure, x-direction diffusion terms are neglected. The resulting equations are:

Continuity

$$\frac{\partial}{\partial x}(h_2h_5u) + \frac{\partial}{\partial y}(h_3h_5v) + \frac{\partial}{\partial z}(h_1h_5w) = 0$$
The energy equation is eliminated by assuming the total enthalpy $h_0$ is constant. The pressure and density are related through the equation of state,

$$p = \gamma / \gamma - 1 \rho \left[ h_0 - \frac{1}{2} (u^2 + v^2 + w^2) \right]$$

A mixing length turbulence model is used, with

$$\mu_T = \mu \left( \frac{1}{h_0} \frac{\partial h}{\partial y} \right) \left( \frac{1}{h_0} \frac{\partial h}{\partial x} \right) \left( \frac{1}{h_0} \frac{\partial h}{\partial z} \right)$$

The mixing length distribution is given by:

$$L_m(y) = 0.008 \tan h \left( \frac{y}{10.09 h_0} \right)$$

The above equations are solved by starting at the ramp tip and marching downstream using an alternating-direction implicit technique. In this study, to avoid resolving the laminar sublayer, wall function boundary conditions were used to compute tangential velocities on all solid surfaces. Details of the solution procedure are published elsewhere.

In the above equations, the mixing coefficients can be computed by any available method. For this study, since only inlets with rectangular cross-sections and parallel sidewalls are considered, $h_3$ and $h_2$ are functions of $x$ and $y$ only, and $h_3 = 1$. The mesh was therefore generated using the two-dimensional method of Anderson.

Test Cases

In order to evaluate the computational procedure described, two test cases were run with the PEPSIS analysis. The first configuration considered was a large scale, variable geometry, mixed compression inlet designed for Mach 3.0 and a Reynolds number per meter of 7.2x10^6 (2.2x10^10 per foot). This test configuration was designated the M3 inlet configuration. It was chosen to study the behavior of the PEPSIS analysis under conditions of high Reynolds number and with a complex crossed shock wave pattern in a largely inviscid core. The second case was a Mach 7.4 hypersonic inlet designed for Mach 3.0 and a Reynolds number per meter of 7.2x10^6 (2.2x10^10 per foot). This test configuration was designated the PS inlet. This case was chosen to study the behavior of the PEPSIS analysis under conditions of strong shock wave interactions with thick boundary layers. In addition, the inlet was used by Knight as a test case for a two-dimensional Navier-Stokes solution. Each inlet configuration was run with two computational mesh distributions, Table I, designated the coarse and medium mesh solutions. The coarse mesh had 20 grid points distributed between the ramp and cowl surfaces while the medium mesh had 45 grid points. The mesh points for both the coarse and medium mesh solutions were packed in the region near the ramp and cowl surfaces to resolve the wall boundary layers. In this study, since only 6 mesh points were used between the inlet sidewall and the symmetry plane, the sidewall boundary layers were not resolved. This did not lead to any computational problems, however. For both inlets, the streamwise step size distribution was the same for the coarse and medium mesh. Later steps in the evaluation procedure will study the effects of varying the spanwise and streamwise mesh size. The total computing times, i.e., CPU plus I/O time, on the UNIVAC 1100/42 are also listed in Table I.

M3 Inlet Configuration

A schematic diagram of the M3 inlet configuration showing the inviscid shock structure is presented in figure (1). The inlet capture area was 1264 cm^2 (160 in^2) and the initial ramp was inclined 7 degrees to the horizontal. The second ramp was inclined 14 degrees to the horizontal while the cowl surface was parallel to the horizontal. The inlet achieves internal compression by means of a crossed shock structure as indicated in figure (1). Calculations were made on the M3 inlet with the ramp, cowl and sidewall bleed regions shown in figure (2). The bleed rates were computed from the data for configuration 80 of reference 12 and are listed in Table II. A comparison between the coarse and medium meshes used in the calculation of the M3 inlet configuration is shown in figure (3). A section upstream on the ramp tip was added to properly resolve the initial ramp shock in the calculations. In the description that follows, $u$ and $v$ are the streamwise and cross-flow velocities in this coordinate system. In addition, the gapwise distance is defined as the distance along a coordinate line from the ramp to the cowl. Comparisons between the coarse and medium mesh solutions are presented in figures (4) - (6). In Figure (4), computed Mach number profiles in the center plane of the M3 inlet have been plotted within the inlet geometry. The coarse mesh solution exhibited substantial numerical noise, particularly in those regions where an oblique shock either formed or was reflected from a solid surface. This can be seen more clearly in figure (5) where the $u$-velocity (streamwise velocity) profiles have been plotted as a function of normalized gap distance across the flow field. Differences in the computed shock wave structure are also evident in figure (5). A more distinct second ramp shock is apparent in the medium mesh solution in addition to an upstream shift in the first ramp shock-boundary layer interaction. The dramatic differences between the coarse and
medium mesh solutions can also be seen in figure (6) which presents the v-velocity (normal velocity) profiles in the M3 inlet center plane as a function of normalized gap distance. The boundary layers on both the ramp and cowl surface were thin relative to the channel gap dimensions so that detailed resolution of these interactions would require additional mesh points. The solution ended just upstream of the throat because of a substantial region of subsonic layer interactions, but both solutions agree. The numerical noise disturbances generated along Mach lines act like weak shock waves. Increasing the mesh density essentially eliminated these numerical disturbances. The medium mesh solution, figure (13a), are propagated along Mach lines and act like weak shock waves. The numerical noise disturbances generated just downstream of both the ramp and cowl shock waves in the coarse mesh solution, figure (13a), are propagated along Mach lines and act like weak shock waves. Increasing the mesh density essentially eliminated these numerical disturbances. The medium mesh solution, figure (13a), had enough grid points to resolve the detailed flow events within the length scale associated with this interaction. The boundary layer buildup along the ramp surface and its interaction with the cowl shock is clearly evident. The static pressure profile development in the center plane of the P8 inlet, figure (14), reveals more clearly that the numerical noise disturbances are propagated along Mach lines in the coarse mesh solution and that increased mesh resolution eliminates these problems. Note also that there is a substantial normal pressure gradient across the ramp and cowl boundary layers within the interaction zone.

Comparisons of the coarse and medium mesh static pressure solutions with measurements in the center plane of the P8 inlet are shown in figure (15). The calculations were performed assuming turbulent boundary layer flow along both the ramp and cowl surfaces while the experimental test had substantial laminar flow on both surfaces.
As a result, the analysis predicted a thicker boundary layer entering both interaction regions which resulted in a slight upstream shift in the shock location. In spite of these differences very good static pressure predictions were achieved with the PEPSIS analysis. The PEPSIS code is presently being updated to include a laminar-turbulent transition capability. It is expected that with thinner boundary layers still better agreement will be achieved. A comparison between the PEPSIS viscous analysis and inviscid calculations is presented in figure (16) and shows a strong ramp shock interaction extending well upstream of the inviscid shock impingement point and a substantial upstream displacement in the cowl shock interaction region.

Concluding Remarks

A series of calculations were performed on two inlet configurations using a three-dimensional viscous flow analysis (PEPSIS) and the following remarks can be made:

1. With proper overall mesh resolution, the PEPSIS analysis is able to define the basic features of high Reynolds number thin boundary layer flows with highly complex shock wave structures in a largely inviscid core.

2. Even when the solution mesh is too coarse to properly resolve the small length scale events associated with thin boundary layer-shock interactions, the overall features of these events can be predicted.

3. With adequate mesh resolution of the shock wave-boundary layer interactions, the PEPSIS analysis predicts the gross physics of this flow event.

4. Depending on the mesh density, the PEPSIS analysis can be used either as a design tool, where the overall inlet flow field is the important factor, or as an analysis tool where the small scale flow events occurring in inlets can be studied.

5. Numerical noise disturbances can develop within the solution which will distort the flow field, but proper mesh resolution can eliminate this problem.

References


### TABLE I. - TEST CASES FOR PEPSIS

<table>
<thead>
<tr>
<th>Test cases</th>
<th>Computational mesh</th>
<th>Computing time&lt;sup&gt;*&lt;/sup&gt; on UNIVAC 1100/42, min</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Distribution</td>
<td>Total</td>
</tr>
<tr>
<td>M3 inlet</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Coarse</td>
<td>20x6x180</td>
<td>21 600</td>
</tr>
<tr>
<td>Medium</td>
<td>45x6x177</td>
<td>47 790</td>
</tr>
<tr>
<td>Medium</td>
<td>45x6x260</td>
<td>70 200</td>
</tr>
<tr>
<td>P8 inlet</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Coarse</td>
<td>20x6x350</td>
<td>42 000</td>
</tr>
<tr>
<td>Medium</td>
<td>45x6x340</td>
<td>91 800</td>
</tr>
</tbody>
</table>

<sup>*</sup>CPU plus I/O time.

### TABLE II. - BLEED RATES FOR M3 INLET CONFIGURATION

<table>
<thead>
<tr>
<th>Bleed zone</th>
<th>$\frac{\dot{m}<em>{BL}}{\dot{m}</em>{\infty}}$</th>
<th>$\frac{v}{u_{ref}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>R&lt;sub&gt;A&lt;/sub&gt;</td>
<td>0.0097</td>
<td>0.00902</td>
</tr>
<tr>
<td>R&lt;sub&gt;B&lt;/sub&gt;</td>
<td>.0053</td>
<td>.00528</td>
</tr>
<tr>
<td>R&lt;sub&gt;C&lt;/sub&gt;</td>
<td>.0193</td>
<td>.00769</td>
</tr>
<tr>
<td>R&lt;sub&gt;D&lt;/sub&gt;</td>
<td>.0214</td>
<td>.00832</td>
</tr>
<tr>
<td>C&lt;sub&gt;A&lt;/sub&gt;</td>
<td>.0120</td>
<td>.00295</td>
</tr>
<tr>
<td>C&lt;sub&gt;B&lt;/sub&gt;</td>
<td>.0699</td>
<td>.00431</td>
</tr>
<tr>
<td>S&lt;sub&gt;A&lt;/sub&gt;</td>
<td>.0120</td>
<td>.00451</td>
</tr>
<tr>
<td>S&lt;sub&gt;B&lt;/sub&gt;</td>
<td>.0147</td>
<td>.00748</td>
</tr>
<tr>
<td>S&lt;sub&gt;C&lt;/sub&gt;</td>
<td>.0176</td>
<td>.01277</td>
</tr>
</tbody>
</table>

$\dot{m}_{\infty} = 7.006 \text{ kg/sec (15.45 lbm/sec)}.$

$u_{ref} = 640.2 \text{ m/sec (2100 ft/sec)}.$
Figure 1. - Schematic diagram of M3 inlet configuration showing inviscid shock structure.

Figure 2. - Bleed regions for M3 inlet configuration. (Distances are model stations from ref. 12.)
Figure 3. - Comparison of coarse and medium mesh used in computation of M3 inlet flow field.

Figure 4. - Mach number profiles in center plane of M3 inlet configuration. $M = 3.0$; $Re = 7.2 \times 10^6/m^2 (2.2 \times 10^9/f.ft)$. 
Figure 5. - Effect of mesh on the u-velocity profiles in center plane of M3 Inlet configuration.

Figure 6. - Effect of mesh on the v-velocity profiles in center plane of M3 Inlet configuration.
Figure 7. - Effect of mesh on the wall static pressure distribution in center plane of M3 inlet configuration.

Figure 8. - Medium mesh flow field solution for M3 inlet configuration at overspeed conditions, $M_a = 3.25$. 

[Graphs and diagrams related to the text are shown here.]
Figure 9. - Schematic diagram of PB hypersonic inlet showing inviscid shock structure.

Figure 10. - Definition of effective cowl leading edge (not to scale).

Figure 11. - Mach number profiles in center plane of PB inlet configuration. $M_m = 7.4$, $Re = 8.9 \times 10^6$ \(m^{-1}(2.7 \times 10^6)\).
Figure 12. Internal Mach number profiles in center plane of P8 inlet configuration. $M_a = 7.4$, $Re = 8.9 \times 10^6/m (2.7 \times 10^5/ft)$.

Figure 13. Effect of mesh on Mach number profiles in center plane of P8 inlet configuration.
Figure 14. - Effect of mesh on static pressure profiles in center plane of P8 inlet configuration.

Figure 15. - Effect of mesh on wall static pressure distribution in center plane of P8 inlet configuration.
Figure 16. - Comparison of medium mesh viscous solution with inviscid analysis for wall static pressure in center plane of P8 inlet configuration.
A three-dimensional fully viscous computer analysis, which retains the viscous nature of the Navier-Stokes equations, was evaluated to determine its usefulness in the design of supersonic inlets. This procedure takes advantage of physical approximations to limit the high computer time and storage associated with complete Navier-Stokes solutions. Computed results are presented for a Mach 3.0 supersonic inlet with bleed and a Mach 7.4 hypersonic inlet. Good agreement was obtained between theory and data for both inlets. Results of a mesh sensitivity study are also shown.