LA-SUB--95-29

RECEIVED
MAR 03 1995
OSTI

"Advanced Aerodynamic Research"

Final Technical Report

LANL Contract No. 5002L0013-9Z

1 December 93

Gordon R. Scriven

G. Newton Freeman

Aero Optics, Inc.

MASTER

DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document.

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, make any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

Outline

A two-dimensional inviscid flow-field code was developed to assess the magnitude and impact of local pressure variations on a curved planar surface. The code uses a unique formulation of the governing conservation equations in curvilinear streamtube coordinates which provides automatic grid generation and step control which are transparent to the user. The code is designed for straightforward extension to mixing-coupled condensing flows.

This report includes 1) descriptions of the physical model and computer code, 2) example flow-field predictions/comparisons for selected test cases, and 3) a plan for code implementation to address and evaluate the condensation phenomena.

Outline

- Code description
- Test-case results
- Implementation plan

Code Capabilities

The code treats both planar and axi-symmetric flow-field geometries with pressure and/or surface boundary conditions. The code is applicable to internal flows through ducts, external flows around bodies, and unbounded jet flows (plumes and wakes). For jet flows, the initial exhaust can be either underexpanded or overexpanded.

Code Capabilities

- **Geometry**
 - Axi-symmetric
 - Two-dimensional
- **Boundary**
 - Pressure
 - Surface
- **Body Flow**
 - External
 - Internal
- **Free Flow**
 - Plume
 - Wake
- **Exhaust**
 - Underexpanded
 - Overexpanded

Flow Computation Options

The code uses a space-marching numerical-integration technique to predict the complete flow-field properties between two impermeable boundaries beginning from a given inflow boundary. The impermeable boundaries are specified either by pressure (free surface) or angle (fixed surface). The inflow boundary is specified by the nozzle flow properties at an arbitrary station downstream of the throat (supersonic flow region). Details of the boundary specifications (user input) are included in subsequent charts.

Flow Computation Options

Option	Boundary Specification	
	Inner	Outer
A	Angle	Pressure
B	Angle	Angle

Compression / Expansion Regions

The surface or pressure boundaries of the nozzle/plume flow exhibit alternate regions of expansion and compression due to turning. A gradual compression turn at the boundary merges into a steep compression (or shock) in the interior. The intersection of multiple compression/expansion waves from opposite boundaries produces a complex region of interaction. The code is designed to predict the corresponding spatial distributions of velocity, pressure, temperature, and density at arbitrary resolution (user option).

Compression / Expansion Regions

Code Structure

The key elements and logic structure of the code are summarized by the indicated sequence of steps. The code input data include exhaust properties, surface geometry, and grid resolution. The input data are processed to generate the grid-point locations and initial-parameter values on the inflow boundary. The numerical computation advances in the downstream direction (determined by the local velocity vector) in accordance with the governing conservation equations (mass, energy, momentum) and boundary conditions (pressure and/or surface). The computation grid and step size are determined automatically by the code. The computed flow-field properties are output at a grid resolution which is commensurate with the specified number of streamtubes.

Code Structure

- **Input:** Specific heat ratio Number of streamtubes
 Exhaust Mach number Streamtube mass ratio
 Exhaust pressure ratio Pressure step control
 Surface boundary geometry Angle step control

- **Initialize:** inflow boundary streamtube geometry

----- Begin computation loop -----

- **Output:** Radial position Mach number
 Axial position Temperature
 Inclination angle Pressure
- **Differential geometry:** angles, distances, areas
- **Differential properties:** velocity, density, temperature, pressure
- **Step size:** streamtube relationships, stability control
- **Incremented variables:** geometry, properties

----- End computation loop -----

Code Input

The code input parameters include exhaust/ambient properties, nozzle/deck geometry, and streamtube grid resolution. For convenience, the exhaust flow is assumed uniform; this assumption can be relaxed at the cost of additional input specifications. For an ideal gas, the flow is fully specified in terms of the specific heat ratio, nozzle Mach number, and ambient-to-exhaust pressure ratio; all flow-field properties (velocity, pressure, temperature, density) are normalized to their initial values. The boundary geometry is specified by angles and angle derivatives with all linear dimensions normalized to the nozzle height at the initial station (e.g., throat or exit). The spatial grid transverse to the flow direction is determined from the user-specified streamtube grid (equal-mass or proportional-mass).

Code Input

Exhaust:	γ	Specific heat ratio
	M (o)	Mach number (uniform)
	P_n (o)	Nozzle pressure ratio
Deck:	θ_n (o)	Nozzle outer angle
	θ_o (o)	Nozzle inner angle
	θ'_o (s)	Surface curvature function
Grid:	N	Number of streamtubes
	α	Streamtube mass ratio

Deck Geometry Specification

The original deck geometry specification was developed as an interim tool for code test and verification pending specification by LANL of the appropriate function and format. The specification used a continuous analytic function to describe the angle derivative $\theta'(s)$. The specification required three parameters to describe the large-scale geometric variations and three additional parameters to describe the small-scale geometric variations.

Deck Geometry Specification

Global	Local
Parameters: $\theta_o, \theta'_o, \theta''_o$	Parameters: z^*, H, λ
$\theta' = \theta'_o + \theta''_o s$ $\theta = \theta_o + \theta'_o s + \theta''_o \frac{s^2}{2}$ $r = \int_0^s \sin \theta ds$ $z = \int_0^s \cos \theta ds$	$h = H \sin^2 \left[\frac{\pi}{\lambda} (z - z^*) \right]$ $\tan \alpha = \frac{H\pi}{\lambda} \sin \left[\frac{2\pi}{\lambda} (z - z^*) \right]$ $\alpha' = 2H \left(\frac{\pi}{\lambda} \right)^2 \cos \left[\frac{2\pi}{\lambda} (z - z^*) \right] \cos^2 \alpha \cos \theta$ $\theta'_{net} = \theta' - \alpha'$

All distances normalized by nozzle exit height.

Deck Surface Geometry: Tabular Input

At the request of LANL, the deck geometry specification was modified to accept tabular input data for deck height (y) as a function of running length (x). The requisite surface angle $\theta_o(s)$ and derivative $\theta'_o(s)$ are then determined by the code.

The deck is described by a one-dimensional input table (x_n, y_n for $1 < n < 200$) where $n = 1$ designates the nozzle exit (lower boundary). The nozzle is described by the lower and upper angles (θ_1 and θ_2) and the height (H).

The algorithm provides continuous values of y and dy/dx by means of a piecewise-uniform approximation for d^2y/dx^2 . The discontinuities in d^2y/dx^2 at the segment transitions should have only moderate impact on the computed flowfield properties.

The aerodynamic effects are independent of scale. Hence, the aerodynamic code uses normalized coordinates: x/H and y/H . Only kinetic effects (subsequent overlay computation) are functions of scale.

Streamtube Initial Geometry

The grid-point locations on the inflow boundary are determined from the user-specified streamtube resolution (number of tubes and mass ratio) and the initial nozzle-boundary angles. Different formulae are used for parallel or nonparallel nozzle boundaries and for planar or axi-symmetric flows. In the figure, the term f_i designates the fractional mass flux in streamtube i from which the radial and axial (r, z) grid point locations are determined.

Streamtube Initial Geometry (given f_i)

Parallel Flow ($\theta_n = 0$)	Nonparallel Flow ($\theta_n > 0$)
$\theta_i = 0$ $\frac{r_i}{r_n} = \begin{cases} f_i & \text{(planar)} \\ f_i^{1/2} & \text{(axisymmetric)} \end{cases}$ $\frac{z_i}{r_n} = 0$	$\theta_i = \begin{cases} f_i \theta_n & \text{(planar)} \\ \cos^{-1} [1 - f_i (1 - \cos \theta_n)] & \text{(axisymmetric)} \end{cases}$ $\frac{r_i}{r_n} = \frac{\sin \theta_i}{\sin \theta_n}$ $\frac{z_i}{r_n} = \frac{\cos \theta_i - \cos \theta_n}{\sin \theta_n}$

Streamtube Mass Allocation

As a convenience, the streamtube mass flux is specified by a simple algorithm based on a constant mass ratio between adjacent tubes. The user specifies only the number of streamtubes and the mass ratio between tubes. The code then computes the corresponding mass flux in each tube.

Streamtube Mass Allocation

f_i = fractional mass flux in streamtube i

n = total number of streamtubes

α = mass flux ratio = f_i / f_{i-1} ($2 < i < n$)

$$\alpha \sum_{i=1}^{n-1} f_i = \sum_{i=2}^n f_i \implies \alpha(1 - f_n) = 1 - f_1$$

$$\frac{f_n}{f_1} = \prod_{i=2}^n \frac{f_i}{f_{i-1}} = \prod_{i=2}^n \alpha = \alpha^{n-1} \implies f_1 = \left(\frac{1 - \alpha}{1 - \alpha^n} \right)$$

$$f_i = \alpha^{i-1} \left(\frac{1 - \alpha}{1 - \alpha^n} \right)$$

Curvilinear Geometry

The streamtube solution is computed on an orthogonal curvilinear grid which is defined by the local streamline direction (θ) and distance (s). The local spacing and curvature of the grid lines is determined by the indicated geometric constraints. These differential equations are advanced together with differential equations of motion to yield an optimum computation grid which is everywhere conformal to the flow field velocity direction (s) and turning direction (n).

Curvilinear Geometry

$$A = \left\{ \begin{array}{l} \Delta \\ \pi (r + r') \Delta \\ \text{planar} \\ \text{axisym} \end{array} \right\}$$

$$\Delta^2 = (r - r')^2 + (z - z')^2$$

$$\frac{dr}{ds} = \sin \theta; \quad \frac{dz}{ds} = \cos \theta$$

$$d\Delta = (ds' + ds) \sin \left(\frac{\theta - \theta'}{2} \right)$$

$$\Delta d \left(\frac{\theta + \theta'}{2} \right) = (ds' - ds) \cos \left(\frac{\theta - \theta'}{2} \right)$$

Governing Equations

In curvilinear streamtube coordinates, the local flow velocity is always parallel to one coordinate while the local diffusion velocity is always parallel to the other coordinate. This natural orthogonal grid yields the simplest possible form of the conservation equations in which shear forces act parallel to the streamlines and transport processes act perpendicular to the streamlines. The resulting numerical solutions are generally more accurate and robust than solutions for non-conformal grids.

Governing Equations (Inviscid)

Mass	$\frac{d\rho}{\rho} + \frac{dv}{v} + \frac{dA}{A} = 0$
------	--

Energy	$\frac{dh}{v^2} + \frac{dv}{v} = 0$
--------	-------------------------------------

State	$\frac{d\rho}{\rho} + \frac{dT}{T} = \frac{dP}{P}$
-------	--

s-Mom	$\frac{dv}{v} + \frac{dP}{\rho v^2} = d\Theta$
-------	--

n-Mom	$\rho v^2 \frac{d\Theta}{ds} + \frac{dP}{dn} = 0$
-------	---

Explicit Solution

The five governing equations can be solved to yield explicit expressions for the differential changes in angle, velocity, density, temperature, and pressure. The angle change is a function of the local pressure gradient normal to the flow direction. The property changes are functions of the local area gradient in the flow direction (as for one-dimensional flow).

Explicit Solution

$$\gamma M^2 \frac{d\theta}{ds} = - \frac{1}{P} \frac{dP}{dn}$$

$$(M^2 - 1) \frac{dv}{v} = \frac{dA}{A} + \gamma M^2 d\theta$$

$$\frac{d\rho}{\rho} = - \frac{dv}{v} - \frac{dA}{A}$$

$$\frac{dT}{T} = - (\gamma - 1) M^2 \frac{dv}{v}$$

$$\frac{dP}{P} = \frac{d\rho}{\rho} + \frac{dT}{T}$$

Streamtube Distance Increments

The local curvature of a given streamtube serves to determine the relative distance increments of the bounding streamlines for each integration step. Thus, the incremental distances for all streamlines can be expressed in terms of the incremental distance for a single reference streamline.

Streamtube Distance Increments

$$\Delta d \left(\frac{\theta + \theta'}{2} \right) = (ds' - ds) \cos \left(\frac{\theta - \theta'}{2} \right)$$

$$\therefore \frac{\Delta}{2} \left(\frac{d\theta}{ds} \frac{ds}{ds'} + \frac{d\theta'}{ds'} \right) = \left(1 - \frac{ds}{ds'} \right) \cos \left(\frac{\theta - \theta'}{2} \right)$$

$$\therefore \frac{ds}{ds'} = \left[\cos \left(\frac{\theta - \theta'}{2} \right) - \frac{\Delta}{2} \frac{d\theta'}{ds'} \right] \left[\cos \left(\frac{\theta - \theta'}{2} \right) + \frac{\Delta}{2} \frac{d\theta}{ds} \right]^{-1}$$

$$\text{Multitube:} \quad \frac{ds_i}{ds_o} = \prod_{i=1}^i \frac{ds_i}{ds_{i-1}}$$

Integration Step Size

Numerical stability and accuracy are controlled at each integration step by limiting the step size according to strict physical criteria. These criteria include the relative pressure change and absolute angle change for each streamtube. The maximum allowable integration step size is determined independently for each streamtube and then weighted according to the previously-determined relative distance increments. The entire flow-field solution then advances by an amount which is commensurate with the most-stressing streamtube and criterion.

Integration Step Size

$ \delta \ln P < \varepsilon_P$	$ \delta \theta < \varepsilon_\theta$
$\delta \ln P = \frac{d \ln P}{ds} \cdot \delta s$ $\therefore \delta s < \varepsilon_P \left \frac{d \ln P}{ds} \right ^{-1}$	$\delta \theta = \frac{d\theta}{ds} \cdot \delta s$ $\delta s < \varepsilon_\theta \left \frac{d\theta}{ds} \right ^{-1}$

- Dual criteria: pressure and curvature
- Use lesser value for each streamtube
- Use minimum value from all streamtubes

Example Test-Case Predictions (Planar Flow)

Oblique shock comparisons / validations

Underexpanded exhaust over flat wall

Underexpanded exhaust over wavy wall

Oblique Shock

Supersonic flow over a wedge produces an attached oblique shock which turns the flow abruptly in a direction parallel to the wedge. The corresponding post-shock flow properties (Mach number, pressure, temperature, etc) are given by a well-known exact solution with tabular results contained in most gas-dynamics textbooks.

The code represents an oblique shock by a continuous turn; the curvature of the turn depends on the streamtube resolution. However, the final properties are independent of the streamtube resolution.

The figure shows a 20-degree wedge in a Mach-3 flow. The oblique-shock turning angle of 37.5 degrees is well-matched by the code predictions.

Oblique Shock

Oblique Shock: Turning

The figure compares the turning angle predicted by the code with the oblique-shock turning angle. Good agreement is shown over a wide range of pressure ratios.

Oblique Shock: Turning

$M_1=3, \gamma=1.4$

Oblique Shock: Temperature

The figure compares the static temperature ratio predicted by the code with the oblique-shock static temperature ratio. Good agreement is shown over a wide range of pressure ratios.

Oblique Shock: Temperature

Oblique Shock: Mach Number

The figure compares the downstream Mach number predicted by the code with the oblique-shock downstream Mach number. Good agreement is shown over a wide range of pressure ratios.

Oblique Shock: Mach Number

Underexpanded Exhaust Over Flat Wall: Computation Grid

Example solutions were obtained for 10 streamtubes of equal mass. The results are displayed in linear spatial coordinates with the vertical scale factor magnified relative to the horizontal scale factor. All linear dimensions are normalized by the nozzle exhaust height.

The figure illustrates the computation grid. The grid is distorted by the different horizontal and vertical scale factors such that it does not appear to be orthogonal (Undistorted plots confirm the orthogonality.) The streamwise increments are shown at intervals of 100 computation steps.

Underexpanded Exhaust Over Flat Wall: Computation Grid

$M=4$, $Pa/Pe=0.8$, $\theta_e=10^\circ$, $\gamma=1.25$

Underexpanded Exhaust Over Flat Wall: Mach Number

The figure illustrates the predicted local Mach number. Maximum and minimum values occur along the flat surface. The highest maximum and lowest minimum occur in the first cell. The length of each cell is proportional to the nozzle height and to the inverse square root of nozzle pressure ratio.

Underexpanded Exhaust Over Flat Wall: Mach Number

$M=4$, $P_a/P_e=0.8$, $\theta_e=10^\circ$, $\gamma=1.25$

Underexpanded Exhaust Over Flat Wall: Pressure

The figure illustrates the predicted local pressure ratio (relative to the nozzle exhaust). Maximum and minimum values occur along the flat surface. The lowest minimum and highest maximum occur in the first cell. The oscillations will continue until pressure equilibrium is achieved in the downstream plume.

Underexpanded Exhaust Over Flat Wall: Pressure

$M=4$, $P_a/P_e=0.8$, $\theta_e=10^\circ$, $\gamma=1.25$

Underexpanded Exhaust Over Flat Wall: Temperature

The figure illustrates the predicted local temperature ratio (relative to the nozzle exhaust). Maximum and minimum values occur along the flat surface. The lowest minimum and highest maximum occur in the first cell. For the given conditions, the predicted range is a factor of two.

Underexpanded Exhaust Over Flat Wall: Temperature

$M=4$, $P_a/P_e=0.8$, $\theta_e=10^\circ$, $\gamma=1.25$

Remaining Issues (Flow Field)

The current flow-field code provides the required capability to predict the magnitude and impact of local pressure variations on a curved planar surface. The current capability is limited to infinite nozzle aspect ratio, continuous deck curvature, and supersonic inviscid flow. Capabilities for mixing and condensation are planned for a subsequent code version.

Remaining Issues (Flow Field)

- Abrupt wall curvature
- Locally subsonic flow
- Finite aspect ratio
- Downstream plume flow
- Coupled viscous mixing
- Condensation code interface

infinite radius of curvature
or
discontinuous angle

Code Implementation Plan

The inviscid code is complete except for anticipated modifications to the deck-geometry interface specification (per LANL). Near-term task requirements include code installation at LANL, predictions/comparisons for Gowadia test cases, and parametric assessment of supersaturation effects. Longer-term task requirements include straightforward code extensions to coupled expansion/mixing/condensation and comprehensive assessment of the underlying engineering design concept. It is anticipated that this work will be accomplished in a follow-on program. A detailed follow-on task plan has been assembled for presentation/evaluation at a Final Review Briefing at LANL.

Code Implementation Plan

- **Near Term (current code)**
 - **Installation and operation at LANL**
 - **Comparison with Gowadia predictions**
 - **Parametric study of local supersaturation**

- **Longer Term (extended code)**
 - **Coupled expansion / mixing code**
 - **Coupled mixing / condensation code**
 - **Engineering concept feasibility study**

Underexpanded Exhaust Over Wavy Wall: Computation Grid

Example solutions were obtained for 10 streamtubes of equal mass. The results are displayed in linear spatial coordinates with the vertical scale factor magnified relative to the horizontal scale factor. All linear dimensions are normalized by the nozzle exhaust height.

The figure illustrates the computation grid. The grid is distorted by the different horizontal and vertical scale factors such that it does not appear to be orthogonal (Undistorted plots confirm the orthogonality.) The streamwise increments are shown at intervals of 100 computation steps.

The exhaust condition is unchanged from the flat-wall example considered previously. The wavy wall is observed to modulate the more-gradual variations induced by the opposing pressure boundary.

Underexpanded Exhaust Over Wavy Wall: Mach Number

The figure illustrates the predicted local Mach number. The results may be compared directly to the flat-wall solution. The multiple expansions and compressions near the wall are observed to result in lower downstream Mach numbers due to the cumulative effects of entropy production.

Underexpanded Exhaust Over Wavy Wall: Pressure

The figure illustrates the predicted local pressure ratio (relative to the nozzle exhaust). The results may be compared directly to the flat-wall solution. Peak compressions are observed at similar locations; however, the magnitudes of the pressure oscillations are more rapidly damped by the wavy wall.

Underexpanded Exhaust Over Wavy Wall: Temperature

The figure illustrates the predicted local temperature ratio (relative to the nozzle exhaust). The results may be compared directly to the flat-wall solution. The multiple expansions/recompressions near the wall are observed to result in higher downstream temperatures due to the cumulative effects of entropy production.