DESCRIPTION AND USER'S MANUAL FOR A PROGRAM TO ANALYZE LOADS AND HEATING ON BODIES SUBJECT TO ROCKET EXHAUST PLUME IMPINGEMENT

HREC/1150-2
LMSC/HREC A791231

LOCKHEED MISSILES & SPACE COMPANY

HUNTSVILLE RESEARCH & ENGINEERING CENTER

HUNTSVILLE RESEARCH PARK

4800 BRADFORD BLVD., HUNTSVILLE, ALABAMA

DESCRIPTION AND USER'S MANUAL FOR A PROGRAM TO ANALYZE LOADS AND HEATING ON BODIES SUBJECT TO ROCKET EXHAUST PLUME IMPINGEMENT

February 1968

Contract NAS8-21150

by

H. W. Butler

APPROVED BY: J. S. Farrior

J. S. Farrior
Resident Manager

FOREWORD

This computer program manual presents the results of work performed by Lockheed's Huntsville Research & Engineering Center for the Propulsion and Vehicle Engineering Laboratory of Marshall Space Flight Center. The work was done under Contract NAS8-21150 and was monitored by James L. Moses of R-P&VE-PTE.

This document constitutes Volume II of a two-part final report for the investigation completed under this contract.

ACKNOWLEDGMENT

The author is grateful to M. L. Blackledge and D. E. Kooker of Lockheed's Thermal Environment Section for their contribution of the continuum and transitional flow heat transfer routines to this computer program. Appreciation is also extended to A. W. Ratliff, Lead Engineer of the Gas Dynamics Group, Propulsion Section, for his overall guidance during the development of this program.

SUMMARY

The computer program which is described in this report was specifically designed to calculate heating rates, surface pressures, forces and moments for a body subjected to rocket exhaust impingement, however, it is capable of analyzing impingement due to any supersonic flow field.

Pressures are calculated in continuum flow using modified Newtonian theory; in free molecular flow using kinetic theory; and in transitional flow using empirical relationships.

Convective heating rates in the continuum or transitional flow regimes are calculated along streamlines using a streamline divergence technique. Heating values for continuum-laminar, continuum-turbulent and transitional flow are obtained at the stagnation point and then distributed over the body according to the streamline divergence. Momentum Reynolds number transition criteria are output in order for the user to choose the most applicable of the three heat rates that are provided. Free molecular heating rates are also calculated at each elemental area from an energy balance-type equation and the decision of which flow regime to consider is simplified by a printout of the applicable flow regime.

The analytical methods used in performing the calculations are described in a report referenced in the document. This report describes the general structure of the program, its routines and the basic capabilities and restrictions. Also, an input guide with an example problem is included.

The program is designed to run on the Univac 1108 digital computer and is written completely in Fortran IV language.

CONTENTS

Section 1
INTRODUCTION

A rocket exhaust constitutes a hostile environment for any adjacent surfaces or spacecraft. Structural loads, heating, radiation and surface erosion are some of the deleterious effects which may occur. In order to predict the severity of the effects, an analytical tool has been developed which will aid in the design of systems where such problems are anticipated.

This document describes a state-of-the-art computer program capable of receiving information concerning the rocket exhaust and determining spacecraft surface pressures, loads and heating. Surface flow conditions on the spacecraft are also presented so that erosion and contamination effects may subsequently be analyzed.

The computer program is described in sufficient detail to provide the reader with an intimate understanding of the program organization, utilization and capabilities. This report describes the actual mechanization of the analysis techniques presented in a companion report, Reference 1.

Section 2
DISCUSSION

The computer program reported here computes pressure loads and heating experienced by a spacecraft or body in a supersonic flow field. A unique vector and matrix mathematical treatment of the body location and its geometrical description is used to streamline the calculational procedure and to simplify interpretation of the results.

The basic program consists of approximately 57 separate routines to perform the vector and matrix operations, surface integration, (described in Reference 1) and perform various tape search and interpolation operations. The nucleus of the program is a finite difference integration scheme. Body surface pressures and heat rates are calculated for each elemental area and forces and torques are integrated over the whole body. Local flow properties used in the program are obtained automatically from a tape generated by Lockheed's Method-of-Characteristics Computer Program (References 2, 3 and 4).

As an aid in the familiarization of the reader with the program's organization and operation, the discussion is divided into five subsections. These subsections are:

- A discussion of the basic capabilities and restrictions of the program
- A brief description of each of the basic routines, in flow chart form, and a brief discussion of the supporting routines
- A definition of each of the common variables
- A discussion of the basic flow of the program
- A detailed discussion of each individual routine used in the program.

Other sections of the report describe the input format, and an application in the form of an example problem is presented.

2.1 BASIC CAPABILITIES

- Axisymmetric or two-dimensional bodies may be treated. The body geometry is described by: (1) conic sections; (b) circular plates; or (c) rectangular plates. Any combination of these three basic shapes may be used up to a maximum of 100 shapes.
- Continuum, transitional or free molecular loads and heating are automatically calculated.
- A printout of surface impact pressure and heating data for each elemental area, or a limited output of force and torque may be selected.
- Options are available which allow the user to consider the body in uniform flow at:
  a. The composite structure origin only;
  b. The origin of each subshape;
  c. At each elemental ring along the stagnation line; and
  d. Each elemental area (see Figure 1 on page 107 for for definition of body geometry).
- The flow field used to determine local properties may be axisymmetric or two-dimensional.
- The program utilizes automatic communication with the flow field tape.
- Dissimilar units of length may be used in the flow field and the impingement description.
- Three different options may be used to calculate local flow properties.

2.2 BASIC RESTRICTIONS

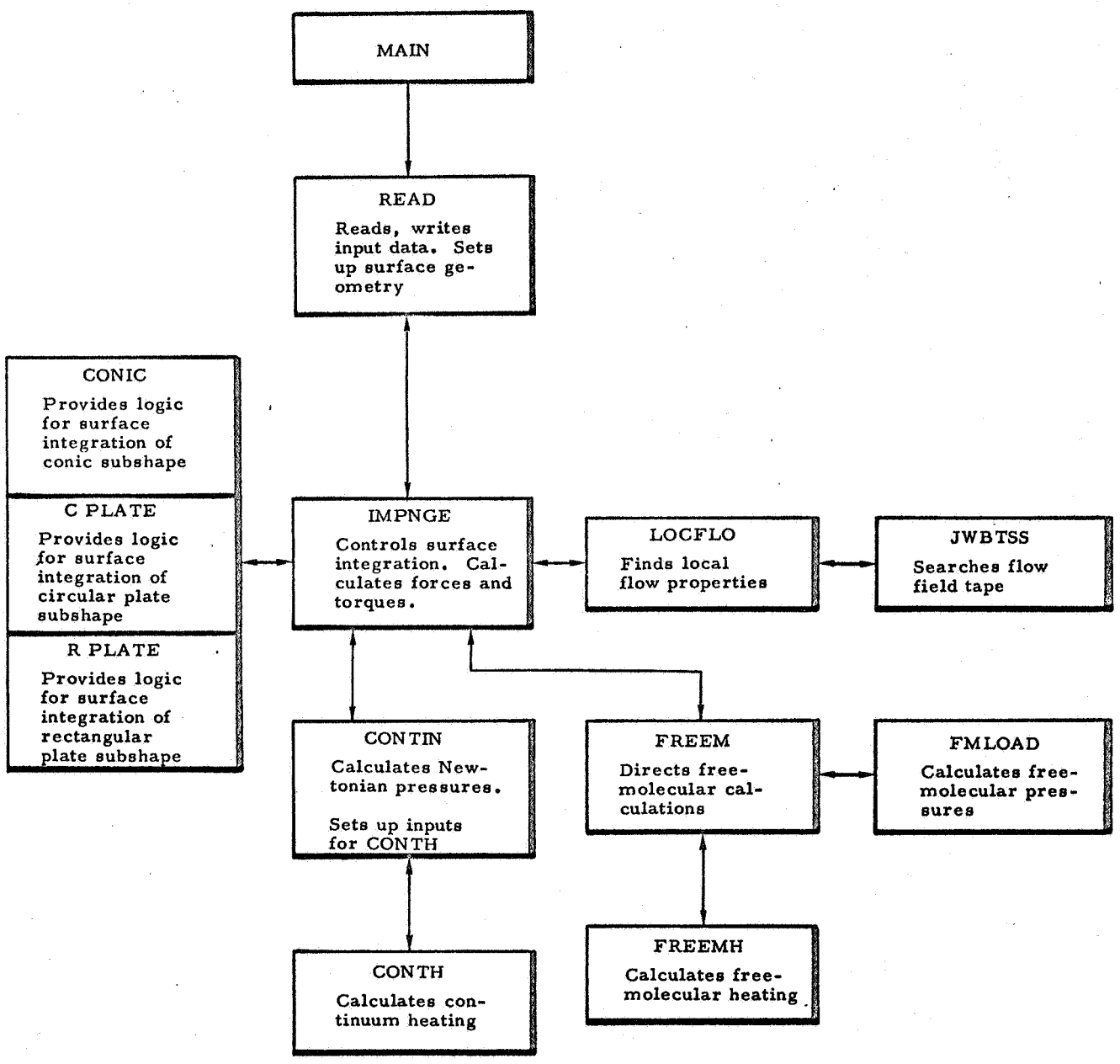
- The composite structure is considered to be a rigid body.
- Continuum heating is limited to axisymmetric body geometry.
- Analytical restrictions are imposed by theoretical methods used in the flow field analysis, heating analysis, force and moment calculations as desribed in Reference 1 through 6.

Section 2.3

BRIEF DESCRIPTION OF ROUTINES

The following subsection contains a brief description of the 57 individual routines which comprise the impingement program. The main routines, which perform the basic logic, are presented in the form of flow charts and supporting routines are described verbally. The routines are grouped into general categories as indicated below.

- Basic Routines
- Vector Manipulation Routines
- General Flow Property Routines
- Shock Calculation Routines
- Tape Manipulation Routines
- Continuum Heating Routines
- Noncontinuum Heat Transfer Routines
- Other Routines



FLOW CHART OF THE BASIC ROUTINES

VECTOR MANIPULATION ROUTINES

Routine	Function
ADDV	adds two vectors
COSDOT	finds included angle between two vectors
CROSS	cross multiplies two vectors
DOT	dot multiplies two vectors
POSTM	transforms a vector from one coordinate system to another
SCALEV	multiplies a scalar times a vector
TRANS	transposes a matrix
VMAG	calculates the magnitude of a vector.

GENERAL FLOW PROPERTIES ROUTINES

Routine	Function
EMOFV	finds Mach number as a function of velocity
POFEM	finds static pressure as a function of Mach number and entropy
RGVOFM	finds velocity as a function of Mach number and entropy. Gas properties are not known prior to entry
RHOFEM	finds density as a function of Mach number and entropy
TABLE	finds local gas properties for an entropy and velocity. It uses a table of gas properties read from flow field tape
TOFEM	finds static temperature as a function of Mach number
TOFV	finds static temperature as a function of velocity
UOFEM	finds Mach angle at a Mach number
UOFV	finds Mach angle as a function of velocity
VOFEM	finds velocity as a function of the Mach number.

SHOCK CALCULATION ROUTINES

Routine	Function
DELTAF	computes the turning angle through an oblique shock wave knowing the shock angle and the upstream Mach number
ENTROP	finds entropy rise across a shock as a function of the shock angle and upstream Mach number
ESHOCK	uses an iterative solution to perform equilibrium shock calculations
WEAK	finds entropy and velocity downstream of a shock.

TAPE MANIPULATION ROUTINES

Routine	Function
JPLGRD	changes the gas properties from MKS units to English units
JPLTAP	reads the gas properties from the flow field tape
READB	reads the boundary points from the flow field tape
READF	reads the left running characteristic lines
TAPMOV	moves the flow field tape past the gas data to the start of the flow field.

CONTINUUM HEATING ROUTINES

Routine	Function
CONTH	controls heating rate calculations and constructs streamlines
PLUME	calculates heating rates at each point on body
PROPTY	calculates gas mixture transport properties as a function of composition and temperature
ONEVAR	interpolation routine

NONCONTINUUM HEAT TRANSFER ROUTINES

Routine	Function
CHTRAN	driver subroutine which decides the applicable Regime (I or II) for the low density flow
ENTPR	calculates the enthalpy function profile in the shock layer
GAMINC	computes the value of the incomplete gamma function
CAC1	computes the slope of the linear portion of the velocity profile in the shock layer
CACUBR	computes the full velocity profile in the shock layer
BLSOL	performs numerical solution of the coupled momentum and energy equations in the shock layer
TRAPZ	used in a trapazoidal integration process.

OTHER ROUTINES

Routine	Function
ERF	computes the error function for routines FMLOAD and FREEMH
FNUDSEN	finds the Knudsen number
HEAD	page ejects the printer and writes the header
ITSUB	controls the iteration for velocity
PRINT	outputs the information at each elemental area
TRANSIT	calculates transitional values of heating and pressure from continuum and free molecular values.

Section 2.4

DEFINITION OF ALL COMMON VARIABLES

This subsection provides a definition of each variable found in a "COMMON" block in the program. The variables are presented as they appear in their respective "COMMON" block with the "COMMON" blocks organized in alphabetical order.

COMMON VARIABLES

Common Name	Variables	Definition	Routines Using
/AMB/	PAMB	ambient pressure of medium outside plume	CONTIN IMPNGE LOCFLO READ
	TAMB	ambient temperature of medium outside plume	
	AB	base area of composite body	
	CD	drag coefficient of composite body	
/ARRY/	U(81)	complete velocity profile in the shock layer	CHTRAN ENTPR
	THET(81)	complete enthalpy function in the shock layer	
	GDO(81)	first approximation of enthapy function in the shock layer	
	THEPR	derivative of the complete enthalpy function at the body surface (Regime II)	
/CONTRL/	IRUN(10)	control flags used by MOC program during plume generation	ESHOCK
	ICON(16)	input controls used by MOC program	
/DUMPCO/	TEMP (15, 320)	local gas properties array obtained from MOC program	JPLTAP JWBTSS READF
	DUMP (8, 200, 3)	local gas properties array obtained from MOC program	
/FLAG/	IFLAG	determines if gas properties have been read into program; 1 – not read in; 2 – have been read in	CONTIN FREEM IMPNGE JWBTSS LOCFLO READ

Common Name	Variables	Definition	Routines Using
/FLAG/ (cont)	JFLAG	indicates if last point was in plume; 2 – last point in plume; 1 – last point not in plume	
	KFLAG	indicates if present point is in plume; 0 – not in plume; 1 – is in plume	
/FORCFO/	FORCE(3)	resultant force on the composite body	CONTIN IMPNGE READ
	TORQUE(3)	resultant torque on the composite body	
	THRUST	(not presently used)	
/FRACTN/	XMF	mole fractions or major constituients	READ PROPTY
/GASCON/	R	local gas constant in plume	CONTH CONTIN DELTAF EMOFP EMOFV ENTROP ESHOCK FMLOAD FNUDSN FREEM JPLGRD LOCFLO POFEM RHOFEM TABLE TOFEM TOFV VOFEM WEAK
	GAMMA	local isentropic exponent in plume	
	TO	total temperature of plume	
	PO	total pressure of plume	
	ASTER	flag which indicates is gas properties are in table or not	
/GASFLO/	PINF	local freestream static pressure in plume	CONTH CONTIN FMLOAD FREEM IMPNGE LOCFLO PRINT
	TINF	local freestream static temperature in plume	
	RHOINF	local freestream static density in plume	

Common Name	Variables	Definition	Routines Using
/GASFLO/ (cont)	H	(not presently used)	
	SPEED	molecular speed ratio	
	CNLOCL	local Knudsen number in plume based on flow gradients	
/HEAT/	PROP (15, 50)	local data array: contains all printed properties for an elemental ring	CONTH CONTIN FREEM IMPNGE PRINT
	AI(8)	local freestream properties; used in free molecular heating calculations	
	AW(8)	local wall properties; used in free molecular heating calculations	
/HEATER/	X	running length from one point to next point	CONTH PLUME
	VLOCAL	local velocity	
	RLOCAL	local density	
	R	radius of ring	
	TLOCAL	local temperature	
	TWALL	wall temperature	
	TOO	total temperature	
	POO	total temperature	
	AMLOCA	local Mach number	
	QLAM	laminar heating rate	
	QTURB	turbulent heating rate	
	RTHETA	momentum thickness Reynolds number	
	QTRAN	heating rate in transition regime	
	DPHI	angular increments between segments on a ring	

Common Name	Variables	Definition	Routines Using
/HEATER/ (cont)	DELVV	velocity growth normal to streamline	
	K	streamline number	
	PI	freestream pressure	
	GAM	isentropic exponent	
	ILBB	control integer equal to zero on first ring	
	XLAM	running wetted length from stagnation point	
/IN/	EPS	low density parameter (EPS = $(\gamma - 1)/2\gamma$), where γ = freestream ratio of specific heats	CHTRAN BLSOL ENTPR CACUBR
	K	square root of the modified low density Reynold's number	
	PR	freestream Prandtl number	
	TWTZ	ratio of wall temperature to freestream stagnation temperature	
	USTR	low density parameter (USTR = $\sqrt{\frac{8\ EPS}{3}}$, where EPS is defined above)	
	F1	reciprocal of USTR	
	F2	low density parameter (F2 = $\frac{K}{\sqrt{2\ USTR}}$, where K and USTR are defined above)	
	DETA	integration step size (DETA = 0.05)	
/INPUT/	IOPT(16)	input option array	CONIC CONTIN FREEM HEAD IMPNGE LOCFLO PRINT READ
	HEADER	title of impingement run	
	NPAGE	page counter	
	LINE	line counter	

Common Name	Variables	Definition	Routines Using
/LOAD/	REFL	composite body reference length	FNUDSN READ
	REFD	composite body reference diameter	
	DIA	molecular diameter of average molecule in plume	
/LOCAL/	RCI(3, 100)	position vectors from composite system origin to subshape origins	CONIC CONTH CONTIN CPLATE FREEM IMPNGE LOCFLO PRINT READ RPLATE
	RI(3)	position vector from subshape origin to surface	
	UNITN(3)	unit normal for current elemental area	
	DA	elemental area	
	DX	axial length of elemental area	
	THETA	impingement angle at elemental area	
	INT	surface integration flag, indicates initial entry and completion of subshape integration	
	ISTAG	subshape number where stagnation point is located	
	XSTAG	axial distance from subshape origin to stagnation point	
/MASSES/	BODYM	(not presently used)	READ
	VEHM	(not presently used)	
	ERTIA(3)	(not presently used)	
/ORIENT/	TEC(3, 3)	transposition matrix from plume exit to composite system origin	IMPNGE LOCFLO READ
	TCE(3, 3)	transpose of TEC(3, 3)	
	TIC (3, 3, 100)	transposition matrix from composite system origin to subshape origins	

Common Name	Variables	Definition	Routines Using
/ORIENT/ (cont)	TCI (3, 3, 100)	transpose of TCI(3, 3, I)	
/OUT/	F(200)	integral of FP (see below)	CHTRAN
	FP(200)	a nondimensional variable representing the velocity profile in the shock layer (used in the low density blunt body solution)	BLSOL CHTRAN ENTPR CACl CACUBR
	TH(200)	a nondimensional variable representing the enthalpy function profile in the shock layer (used in the low density blunt body solution)	
	THP	derivative of the nondimensional enthalpy function at the body surface (Regime I)	
/POSVEC/	REC(3)	position vector from nozzle exit to composite system origin	LOCFLO READ
	RECD(3)	(not presently used)	
	WE(3)	(not presently used)	
	WDE(3)	(not presently used)	
	WDC(3)	(not presently used)	
	XTE	transfer distance from plume origin to exit plane of plume nozzle	
/PQOA/	PC	continuum impact pressure	CONTH
	QC	continuum heat flux	CONTIN FMLOAD
	PF	free molecular impact pressure	FREEM IMPNGE PRINT
	PW	impact pressure at body surface, may be continuum, free molecular, or combination	
	QW	heat flux at body surface, may be continuum, or free molecular	

Common Name	Variables	Definition	Routines Using
/QRCOM/	QRC(3)	velocity of flow in composite system	CONTH CONTIN FMLOAD FREEM IMPNGE LOCFLO PRINT
	QRCG(3)	velocity of flow at composite system origin in composite system	
	AKN	Knudsen number at body surface based on computed characteristic dimension	
	QRI(3)	velocity of flow in subshape systems	
	EM	local Mach number of flow	
	V	local scalar velocity of flow	
	S	molecular speed of flow	
/RAPA/	SMK	low density variable equal to $2.0\ PR/3.0\left(\sqrt{1.0 + \frac{4.0}{K^2}} - 1.0\right)$	CHTRAN ENTPR CAC1 CACUBR
	SMKP	low density variable equal to SMK/PR	
	TR	ratio of wall temperature to freestream stagnation temperature (note: TR = TWTZ)	
	CN	function defined in subroutine ENTPR	
	C1	slope of the linear velocity profile used as a first approximation (see discussion on CAC1)	
	WO	function defined in subroutine ENTPR	
	W1	function defined in subroutine ENTPR	
	FACUO	function define in subroutine ENTPR	
	BO	function define in subroutine ENTPR	
	CO	function define in subroutine ENTPR	

Common Name	Variables	Definition	Routines Using
/RAPA/ (cont)	DO	function define in subroutine ENTPR	
	BKP	low density variable equal to (SMK - SMKP)	
/RING/	DXXX	distance from stagnation point to center of ring	CONIC CONTH READ
	DELPHI	angular constant which orients initial angular step off stagnation line	
/STAGCO/	AK	thermal conductivity	CONTH FMLOAD FREEM READ
	EMU	viscosity	
	CP	specific heat at constant pressure	
	RE	Reynolds number	
	HO	total enthalpy	
	HCHEM	total chemical enthalpy	
	TW	wall temperature	
	CPMAX	maximum pressure coefficient	
	JLB	heating calculation flag: 0 – first ring; 1 – successive rings	
/SURFO/	IDSURF (100)	surface identification flag: 1 – conic; 2 – circular plate; 3 – flat plate	CONIC CONTH CONTIN CPLATE FREEM IMPNGE LOCFLO PRINT READ RPLATE
	COEF (7, 100)	coefficients of subshape surface equations	
	ISURE	surface number flag	
	MXSURF	total number of subshapes	
	NX	number of axial increments in each conic subshape	
	NPHI	number of angular increments in each elemental ring	
	IORNT	orientation flag: 0 – integrate from front to back; 1 – integrate from back to front	

Common Name	Variables	Definition	Routines Using
/SURFO/ (cont)	NS	surface shading flag: 0 – surface is shaded; 1 – surface not shaded	
	INX	number of present axial increment being calculated	
	IPHI	number of present angular increment in conic or circular plate subshape	
	IX	number of present X increment in rectangular plate subshape	
	IY	number of present Y increment in rectangular plate subshape	
	IR	number of present radial increment in circular plate subshape	
	PHIROT	angle from positive Y axis to stagnation line	
	ICONIC (100)	not presently used	
/TABCOM/	STAB(9)	entropy value for each entropy table	JPLGRD JPLTAP RGVOFM TABLE
	IVTAB(9)	number of velocity cuts in each entropy table	
	TAB (9, 13, 5)	local properties at each velocity cut in each entropy table	
	XSI (9, 13, 2)	not presently used	
/TAPEFO/	ALPHA(4)	identification of propellant case	JPLGRD JPLTAP RGVOFM TABLE
	IS	number of entropy cuts in the gas table	
/TIMEFO/	DT	not presently used	IMPNGE READ
	TIME	not presently used	
	TMAX	not presently used	

Common Name	Variables	Definition	Routines Using
	INTCUT	not presently used	
	ISTEP	not presently used	
/UNIT/	TUNIT	constant to make plume and impingement units compatable	CONTH IMPNGE LOCFLO READ
	PUNIT	constant to make plume and impingement units compatable	
/UNITV/	UNITIC(3)	unit vector of composite system x axis	CONTH READ
	UNITJC(3)	unit vector of composite system y axis	
	UNITKC(3)	unit vector of composite system z axis	

Section 2.5

BASIC FLOW OF PROGRAM

The following section describes verbally a typical course the program logic might take in arriving at a solution. A description of the program "flow" is helpful in understanding the general sequence of events.

MAIN PROGRAM

Calls READ

READ

(1) Read and writes inputs

(2) Forms transformation matrices TEC + TCE

(3) Forms transformation matrices TIC and TCI

(4) Calls IMPNGE for surface conditions

IMPNGE

(1) Calls LOCFLO for gas properties at the origin of the composite body

LOCFLO

(1) Transforms the origin of the body from the exit system to the flowfield system

(2) Transforms the origin into a two-dimensional plane

(3) Changes the description for the position of the point to units used in the flow field

(4) Calls JWBTSS to find the flow properties at the composite system origin

JWBTSS

(1) Uses JPLGRD and JPLTAP to read flow field tape

(2) Boundary points of the plume are read in and are checked against the body point location

(3) If boundary check is passed, a search is made to bracket the point

(4) If the point is located, a linear interpolation is made for the Mach number, the flow angle, and the entropy

(5) Control is then returned to routine LOCFLO

LOCFLO

(1) If the composite origin is outside the flowfield, this routine assigns ambient conditions to the gas properties.

(2) If the composite origin is within the flowfield, RGVOFM, TABLE and POFEM are called to calculate local flow properties in the flowfield.

IMPNGE

(1) Calculates the angle PHIROT from the velocity vector QRC. (PHIROT is the angle from the Y axis in the YZ plane to the stagnation stream line)

(2) Calculates the direction of integration along the X axis

(3) Calls PRINT to print heading information

(4) If option for uniform flow at the subshape was chosen, LOCFLO is again called to get the gas properties at the subshape origins

(5) Calls CONIC, RPLATE or CPLATE to get the elemental area and its position (if body is axisymmetric and continuum heating is desired, initial entry into conic is at stagnation point

CONIC

(1) Adjusts logic to start integration at stagnation point

(2) Calculates initial distance from stagnation point to center of elemental ring

(3) Calculates the elemental angular step and the elemental step in the X direction

(4) Takes the first X step and the first angular step

(5) Calculates the area of the elemental segment and the unit normal

(6) Returns control to IMPNGE

IMPNGE

(1) If the option for uniform flow at the stagnation line or the elemental area was chosen, LOCFLO is called to get the gas properties at this point

(2) The body Knudsen number is retrieved from FNUDSN

(3) Calls CONTIN

CONTIN

(1) Tests for shadowing

(2) Calculates the Newtonian impact pressure

(3) Tests for shock – If there is a shock, ESHOCK is used to calculate properties downstream of the shock

(4) Determines gas properties at the surface of the body based on one of the three options chosen and loads them in the PROP array for CONTH

(5) Calls CONTH for continuum heat rates

CONTH

(1) Calculates stagnation point heat flux for continuum laminar and turbulent flow

(2) Calls CHTRAN to obtain stagnation point heat flux to a 1 ft. hemisphere assuming transitional flow

(3) Returns control to CONTIN

CONTIN

(1) Returns control to IMPNGE

IMPNGE

(1) Calls FREEM

FREEM

(1) Calls FMLOAD for free molecular impact pressure

(2) Prepares input values for FREEMH

(3) Calls FREEMH for free molecular heat rates

(4) Returns control to IMPNGE

IMPNGE

(1) Checks the Knudsen number to determine flow regime

(2) Compares flow Knudsen number to body Knudsen number and chooses the one which is closer to free moleculare flow

(3) If in the transitional regime, TRNSIT is called to calculate the pressure from the values provided by CONTIN and FREEMH using an empirical formula

(4) Calls PRINT to write information at the elemental area

(5) Calculates the elemental force and torque

(6) Sums the elemental forces and torques for each subshape

(7) Continues to operate in this loop until the subshape is integrated

(8) When the subshape integration is completed, forces and torques are transformed to the composite system

(9) Operation is returned to the loop for the next subshape

(10) After the body is completely integrated, in the forward direction, a flag is set and CONIC is reentered at the stagnation point. The remainder of the body, from the stagnation point forward, is then integrated

(11) Total force and torque is calculated for the entire body

(12) Control is returned to READ

READ

(1) Terminates the calculation

Section 2.6

DETAILED DISCUSSION OF THE INDIVIDUAL ROUTINES

This subsection contains a detailed description of each routine used in the program.

Described are:

- Function (if applicable) of each routine
- Calling sequence
- Common blocks and other routines used, and
- The method used in performing the routine functions.

For user convenience, the routines are organized alphabetically.

SUBROUTINE NAME: ADDV

DESCRIPTION

This subroutine adds two vectors, one of which is multiplied by a scalar before the addition. The resulting vector is returned.

CALLING SEQUENCE

CALL ADDV (VA, VB, VC, F)

VA and VB - Vectors which are added

VC - Vector sum of VA and VB

F - Scalar

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

The scalar F is multiplied by each component of vector VB.

VC = VA + F * VB

SUBROUTINE NAME: BLSOL

DESCRIPTION

This subroutine determines a numerical solution to two coupled ordinary second order linear differential equations, one representing a velocity profile and the other representing an enthalpy function profile in the stagnation region of a blunt body in a low density flow. This routine is used when the value of the freestream parameters indicates the regime of validity is Regime I. (See Reference 1).

CALLING SEQUENCE

CALL BLSOL

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/IN/
COMMON/OUT/
FUNCTION TRAPZ

METHOD OF SOLUTION

The two coupled ordinary second order linear differential equations result from a similarity transformation applied to the original partial differential equations representing conservation of momentum and energy in the stagnation region. The two coupled ordinary differential equations are solved with an iteration technique where an enthalpy function profile is guessed initially. Then this profile is used in the momentum equation to obtain a solution for the velocity profile. This velocity profile is substituted back into the energy equation to see if the resultant enthalpy function profile function matches the initial guess. If not, the procedure is repeated until the velocity profile function and the enthalpy profile function have converged and satisfied the boundary conditions behind the shock wave and at the body surface.

SUBROUTINE NAME: CAC1

DESCRIPTION

This subroutine calculates the slope, C1, of a linear velocity profile in the stagnation region shock layer of a blunt body in a low density flow. This linear velocity profile is used as a first approximation to the full velocity profile in the shock layer. This routine is used when the value of the freestream parameters indicates the regime of validity is Regime II. (See Reference 1).

CALLING SEQUENCE

```
CALL CAC1
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/IN/
COMMON/RAPA/
GAMINC

METHOD OF SOLUTION

The value of the slope, C1, of the linear velocity profile is calculated from an analytical expression resulting from the solution of the first order perturbation problem based on the momentum equation and the energy equation in the stagnation shock layer.

SUBROUTINE NAME: CACUBR

DESCRIPTION

This subroutine calculates the value of the complete velocity function at a given point in the stagnation region shock layer of a blunt body in a low density flow. This routine is used when the value of the freestream parameters indicates the regime of validity is Regime II. (See Reference 1).

CALLING SEQUENCE

CALL CACUBR (Z, UBAR)

Z = zeta = nondimensional normal coordinate in the stagnation shock layer

UBAR = value of the complete velocity function

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/IN/
COMMON/RAPA/
GAMINC

METHOD OF SOLUTION

The value of the complete velocity function UBAR at a given point in the stagnation point shock layer, Z, is calculated from an analytical expression resulting from the solution of the second order perturbation problem based on the momentum and energy equation in the shock layer. This analytical expression requires the value of C1, the slope of first approximation linear velocity profile, which comes in through COMMON/RAPA/

SUBROUTINE NAME: CHTRAN

DESCRIPTION

This subroutine is a driver for the other low density heat transfer routines. It decides which type of calculation procedure is to be used (Regime I or Regime II) based on freestream properties. It computes the final value of heat transfer coefficient.

CALLING SEQUENCE

CALL CHTRAN (SQK, EPS, PR, TR, CH)

$$SQK = K^2 = \frac{P_\infty a}{\mu_\infty U_\infty C}$$ (modified low density Reynolds number)

where P_∞ = freestream pressure

a = local body nose radius

μ_∞ = freestream viscosity

U_∞ = freestream velocity

C = constant in the relation $\frac{\mu}{\mu_\infty} = C \frac{T}{T_\infty}$

$EPS = \frac{(\gamma - 1.0)}{2\gamma}$ where γ = freestream ratio of specific heats

PR = Prandtl number (freestream)

TR = ratio of body surface temperature to freestream stagnation temperature

CH = heat transfer coefficient

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/IN/
COMMON/OUT/
COMMON/RAPA/
COMMON/ARRY/
ENTPR
BLSOL

METHOD OF SOLUTION

(Not applicable)

SUBROUTINE NAME: CONIC

DESCRIPTION

This routine locates the stagnation point on the composite body and provides the necessary logic to divide any conic subshape into elemental areas for a surface integration.

CALLING SEQUENCE

CALL CONIC

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/INPUT/
COMMON/SURFO/
COMMON/RING/
SCALEV
VMAG

METHOD OF SOLUTION

The routine divides the subshape into the specified number of axial segments or rings of equal width. The rings are then subdivided into equal angular segments. The areas, unit normals and position vectors (RI) are calculated for each segment.

The initial entry into CONIC is with the subshape on which the stagnation point is located and the axial distance from this subshape origin to the stagnation point. Integration proceeds rearward from the stagnation point to the end of the body. When the rearward integration is complete, the logic provides a return to the stagnation point and the integration proceeds in a forward direction to the front of the body.

SUBROUTINE NAME: CONTH

DESCRIPTION

This subroutine provides the streamline tracing technique along which continuum heating rates are calculated over bodies immersed in a real gas flow field. The basic body shapes must be represented by axisymmetric surfaces, but a large variety of shapes may be considered.

CALLING SEQUENCE

```
CALL CONTH
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/INPUT/
COMMON/UNIT/
COMMON/LOCAL/
COMMON/SURFO/
COMMON/UNITV/
COMMON/STAGCO/
COMMON/HEAT/
COMMON/PQOA/
COMMON/GASCON/
COMMON/GASFLO/
COMMON/QRCOM/
COMMON/RING/
COMMON/HEATER/
CHTRAN
PLUME

METHOD OF SOLUTION

Local flow properties are supplied to the subroutine as a function of angular increments from the stagnation line of the body. The spreading of the flow streamlines is calculated from the stagnation point. These streamlines are followed down the body and their location specified at the center of each elemental ring. At each step, heating rates are calculated by calling subroutine PLUME and supplied to the main program in an array. The heating rates output are laminar, turbulent and transitional. A transition criteria is also output so that the user may select the appropriate heating rate.

SUBROUTINE NAME: CONTIN

DESCRIPTION

This routine calculates Newtonian impact pressure at each elemental area. It also prepares local surface properties at each elemental area for subroutine CONTH.

CALLING SEQUENCE

```
CALL CONTIN
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/AMB/	CONTH
COMMON/FORCFO/	EMOFV
COMMON/GASCON/	ESHOCK
COMMON/GASFLO/	POFEM
COMMON/HEAT/	RHOFEM
COMMON/INPUT/	TABLE
COMMON/PQOA/	TOFEM
COMMON/QRCOM/	ITSUB
COMMON/SURFO/	EMOFP
COMMON/STAGCO/	VOFEM
COMMON/FLAG/	UOFV
COMMON/LOCAL/	VMAG

METHOD OF SOLUTION

The impact pressure is calculated for each elemental area using the Newtonian equation listed on the next page.

$$P = P_\infty (1+\gamma_\infty M_\infty^2 \sin^2(\theta))$$

where

P is the Newtonian impact pressure

P_∞, γ_∞, M_∞ are the freestream static pressure, isentropic exponent, and Mach number

θ is the impingement angle.

Local flow properties at the body surface are calculated and stored in an array for use by the continuum heating routine CONTH. These local flow properties are calculated based on one of three input options. Each of these options are described below.

- Option 1: The first option is a Newtonian flow assumption in which the flow goes through an oblique shock parallel to the surface. The normal component of the velocity is stagnated, and oblique shock relations are used to find the local flow properties.

- Option 2: The second assumption is a modified Newtonian approach in which the local properties are obtained as in the first option; however, the shock angle is obtained in an interative solution of the downstream local pressure. This local pressure is obtained from the relationship

$$C_p = \frac{P_L - P_\infty}{1/2\ \rho_\infty V^2}$$

$$C_p = C_{p_{max}} \sin^2\theta$$

where C_P is the pressure coefficient, and $C_{p_{max}}$ is an input quantity

- Option 3: The third option is an isentropic expansion method in which the local pressure is again obtained as in the first option. Stagnation properties are obtained behind a normal shock, and an isentropic expansion to the local pressure is performed. Based on this expansion, other necessary local properties are calculated.

The user must decide, based on the character of the flow field being evaluated and the application of the heat transfer data, which of these three options is most representative for local flow conditions. When there is any doubt, the most generally used assumption is option 1, Newtonian flow.

FUNCTION NAME: COS DOT

DESCRIPTION

This function finds the included angle between two vectors.

CALLING SEQUENCE

FUNCTION = COS DOT (VA, VB)

where VA and VB are any two vectors.

UTILITY ROUTINES AND COMMON REFERENCES

COMMON-None

DOT

VMAG

METHOD OF SOLUTION

The dot product of the two vectors is taken,

$$\vec{VA} \text{ DOT } \vec{VB} = /VA//VB/\cos\theta$$

Then from the relation

$$\cos\theta = \frac{\vec{VA} \text{ DOT } \vec{VB}}{/\vec{VA}//\vec{VB}/}$$

and

$$\theta = \tan^{-1}\left(\frac{\sqrt{1 + \cos^2\theta}}{\cos\theta}\right) ,$$

The included angle θ is calculated.

SUBROUTINE NAME: CPLATE

DESCRIPTION

This routine divides the surface of a circular plate or annulus into elemental areas and provides the logic for a surface integration.

CALLING SEQUENCE

```
CALL CPLATE
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/LOCAL/
COMMON/SURFO/
UTILITY ROUTINES-None

METHOD OF SOLUTION

The plate is first divided angularly, and then these angular segments are divided radially from the center to the outside edge. The area, unit normal and position vector, of each segment is then calculated.

SUBROUTINE NAME: CROSS

DESCRIPTION

This subroutine calculates the cross product of two vectors.

CALLING SEQUENCE

CALL CROSS (VA, VB, VC)

where VA and VB are any two vectors and VC is the cross product.

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

Vector VA is crossed into the vector VB and the resultant vector VC is returned.

FUNCTION NAME: DELTAF

DESCRIPTION

This function computes the turning angle through an oblique shock wave knowing the shock angle and the upstream Mach number.

CALLING SEQUENCE

DELTA = DELTAF (EPS, EM)

where (DELTA) the turning angle is found from the shock angle (EPS) and the upstream Mach number (EM). NOTE: The appropriate values of the ratio of specific heats must be in common corresponding to the input Mach number and the upstream entropy value (see Subroutine Table).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
UTILITY - None

METHOD OF SOLUTION

The oblique shock relations are solved for the turning angle using the relations;

$$\delta = \epsilon - \tan^{-1}\left\{\tan\epsilon\left(\frac{1}{M^2\sin^2\epsilon} + \frac{\gamma - 1}{2}\right)\left(\frac{2}{\gamma + 1}\right)\right\}$$

FUNCTION NAME: DOT

DESCRIPTION

This function calculates the dot product of two vectors and returns the result.

CALLING SEQUENCE

FUNCTION = DOT (VA, VB)

where VA and VB are any two vectors.

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

Vector VA is dotted into vector VB and the resultant is a scalar returned as DOT.

FUNCTION NAME: EMOFP

DESCRIPTION

This routine computes the local Mach number as a function of local pressure (static) and the local entropy value.

CALLING SEQUENCE

EM = EMOFP (P, S)

where (EM) is the resultant Mach number found from the pressure (P) and entropy (S). NOTE: The appropriate values of the gas properties must be stored in common upon entry to this routine (see Subroutine Table).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
UTILITY - None

METHOD OF SOLUTION

Perfect gas relationships (thermally perfect) are used to find the Mach number.

$$M = \sqrt{\left[\left(\frac{p_o e^{-S/R}}{p}\right)^{\frac{\gamma-1}{\gamma}} - 1\right]\frac{2}{\gamma - 1}}$$

FUNCTION NAME: EMOFV

DESCRIPTION

This routine finds Mach number as a function of velocity.

CALLING SEQUENCE

EM = EMOFV (V)

where (EM) is the local Mach number which is found as a function of (V) the local velocity. NOTE: The appropriate values of the gas properties must be stored in common upon entry to this routine.

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
TOFV

METHOD OF SOLUTION

Thermally perfect gas relationships are used to find the Mach number.

$$M = \sqrt{\left(\frac{T_o}{T} - 1\right)\left(\frac{2}{\gamma - 1}\right)}$$

SUBROUTINE NAME: ENTPR

DESCRIPTION

This subroutine calculates the enthalpy function profile in the stagnation region of a blunt body assuming a low density oncoming flow field. This routine is used when the value of the freestream parameters indicates the regime of validity is Regime II. (See complete discussion in Reference 1).

CALLING SEQUENCE

CALL ENTPR (SQK, EPS, PR, TR)
(see definitions under CHTRAN)

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/IN/	GAMINC
COMMON/RAPA/	CAC1
COMMON/ARRY/	CACUBR

METHOD OF SOLUTION

The enthalpy function profile across the shock layer is based on the analytical solution described in Reference 1. Since the energy equation and the momentum equation are coupled, ENTPR calls CAC1 and then CACUBR to obtain the velocity profile first. This information is used in the equations for enthalpy function where a numerical integration must be performed to obtain the final answer.

FUNCTION NAME: ENTROP

DESCRIPTION

This routine utilizes the oblique shock relations to find the entropy rise across a shock as a function of the shock angle and the upstream Mach number.

CALLING SEQUENCE

DS = ENTROP (EPS, EM)

where (DS) is the entropy rise across the shock and is a function of the shock angle (EPS) and the upstream Mach number (EM). NOTE: The appropriate values of the gas properties must be stored in common upon entry to this routine (see Subroutine Table).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
UTILITY - None

METHOD OF SOLUTION

The oblique shock relations are employed to find the entropy rise across the shock.

$$ds = \frac{R}{\gamma - 1}\left\{\ell n\left[\frac{(2\gamma M^2 \sin^2\epsilon - (\gamma - 1))}{\gamma + 1}\right] + \gamma\, \ell n\left[\frac{\tan(\epsilon - \delta)}{\tan\epsilon}\right]\right\}$$

FUNCTION NAME: ERF

DESCRIPTION

This function computes the error function.

CALLING SEQUENCE

ERFZ = ERF (Z)

where ERFZ is the error function of Z

UTILITY ROUTINES AND COMMON REFERENCES

NONE

METHOD OF SOLUTION

The error function of Z is calculated using a curve fit of the general equation.

$$\text{ERF}(Z) = \frac{2}{\sqrt{\pi}} \int_0^Z e^{-Y^2}\, dy$$

SUBROUTINE NAME: ESHOCK

DESCRIPTION:

This subroutine employes an iterative solution to perform the equilibrium shock calculations for a real or ideal gas. The real and ideal gas calculations are similar, the difference being that an ideal gas case converges on the first iteration.

CALLING SEQUENCE

CALL ESHOCK (S1, V1, EP, DELTA, S2, V2)

where the input properties are (S1, V1) the upstream entropy and velocity and (EP) the shock angle. The subroutine returns with (DELTA), the turning angle and (S2, V2), the downstream entropy and velocity.

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/CONTRL/
COMMON/GASCON/

TABLE	RHOFEM	WEAK
EMOFV	ENTROP	
POFEM	DELTAF	

METHOD OF SOLUTION

The equation for conservation of mass through a shock wave and the two independent equations for momentum tangential and normal to the shock wave are rearranged. The rearrangement allows for expressing the equation as functions of four unknowns:

1. ϵ - the shock angle
2. δ - the turning angle
3. S2 - the entropy level downstream of the shock
4. q_2 - the velocity downstream of the shock

One of the unknowns, ϵ, is taken as an independent variable and an iterative solution employed to solve for the other three.

For a more detailed description of the method of solution and a derivation of the equations used, refer to Section 7 of Reference 2.

SUBROUTINE NAME: FMLOAD

DESCRIPTION

FMLOAD calculates the force per unit area (dF/dA) for an elemental area in free molecular flow.

CALLING SEQUENCE

```
CALL FMLOAD
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/LOCAL/
COMMON/QRCOM/
COMMON/SURFO/
COMMON/PQOA/
COMMON/GASCON/
COMMON/GASFLO/
COMMON/STAGCO/
ERF
VMAG

METHOD OF SOLUTION

dF/dA is found from the differential equation for an elemental area presented by Sentman in Reference 5. This equation and its variables can be found also in Reference 1.

FUNCTION NAME: FNUDSN

DESCRIPTION

This function calculates Knudsen numbers for determining the local flow regime.

CALLING SEQUENCE

AKN = FNUDSN (THETA, RHO)

THETA = Flow angle at the surface

RHO = density of flow

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
COMMON/LOAD/
UTILITY – none

METHOD OF SOLUTION

An effective reference length (perpendicular to the direction of flow) is calculated from input reference values. Knudsen number is then calculated using the equation

$$K_s = \frac{MFP}{REF}$$

where REF is the effective reference length and MFP is the mean free path of molecular travel. (See Reference 1).

SUBROUTINE NAME: FREEM

DESCRIPTION

This routine calls subroutine FMLOAD and FREEMH for free molecular pressure and heating. It sets inputs for FREEMH.

CALLING SEQUENCE

CALL FREEM

UTILITY ROUTINE AND COMMON REFERENCES

COMMON/GASCON/	FMLOAD
COMMON/GASFLO/	FREEMH
COMMON/HEAT/	
COMMON/LOCAL/	
COMMON/PQOA/	
COMMON/QRCOM/	
COMMON/STAGCO/	
COMMON/SURFO/	
COMMON/FLAG/	
COMMON/INPUT/	

METHOD OF SOLUTION

Not applicable

SUBROUTINE NAME: FREEMH

DESCRIPTION

This subroutine computes the free molecular heat transfer rates in arbitrary gases. The heat rates can be calculated for either front or back side heating at angle-of-attack.

CALLING SEQUENCE

CALL FREEMH (AI, AW, S, THETA, ALPHA, NS, DQODA)

where (AI, AW) are arrays consisting of five variables each. The variables in the AI array are based on the incident flow and the variables in the AW array are based on the reflected flow. These variables are:

1. enthalpy
2. pressure
3. density
4. gas constant
5. temperature

THETA is the angle between the freestream flow direction and the positive y direction of the surface.

S is the molecular speed ratio.

ALPHA is the thermal accommodation coefficient.

NS selects the side of the surface to consider.

DQODA is the free molecular heat transfer rate.

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/CHECK
ERF

METHOD OF SOLUTION

The free molecular heat transfer rate is calculated for a unit area of a surface at angle-of attack in arbitrary gases. The properties of the incident flow are calculated prior to entering FREEMH. Parameters common to both front and back side heating are calculated. Those parameters which are different are calculated depending on the side to be considered (NS). The heat rate is then calculated.

Additional information on the development of this subroutine can be found in Appendix D of Reference 1.

SUBROUTINE NAME: GAMINC

DESCRIPTION

This subroutine calculates the value of the incomplete gamma function using an infinite series. Since no asymptotic approximation was made, the series will accurately calculate the value of the incomplete gamma function for any value of the variable X and any order of incompleteness A.

CALLING SEQUENCE

CALL GAMINC (X, A, GAMI)
X = arguement of the function
A = order of the gamma function
GAMI = value of the incomplete gamma function

UTILITY ROUTINES AND COMMON REFERENCES

none – routine is self contained

METHOD OF SOLUTION

Successive terms in the following infinite series are calculated until the n^{th} term makes a negligible contribution

$$\Gamma_A(X) = X^A \sum_{n=0} \frac{(-X)^n}{(A+n)\, n!}$$

where $\Gamma_A(X)$ is the incomplete gamma function of order A.

SUBROUTINE NAME: HEAD

DESCRIPTION

This subroutine indexes pages and writes case, and page numbers on each page of printout. It also writes a problem title on each page.

CALLING SEQUENCE

CALL HEAD

UTILITY ROUTINE AND COMMON REFERENCES

COMMON/INPUT/
UTILITY-None

METHOD OF SOLUTION

Not applicable

SUBROUTINE NAME: IMPNGE

DESCRIPTION

This subroutine provides surface integration control for the program. It calculates and sums forces and torques and writes them at the end of each trajectory step.

CALLING SEQUENCE

```
CALL IMPNGE
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/AMB/
COMMON/FORCFO/
COMMON/GASFLO/
COMMON/INPUT/
COMMON/LOCAL/
COMMON/ORIENT/
COMMON/PQOA/
COMMON/QRCOM/
COMMON/SURFO/
COMMON/TIMEFO/
COMMON/UNIT/
COMMON/FLAG/

ADDV
CONIC
CONTIN
COSDOT
CPLATE
CROSS
DOT
FNUDSEN
FREEM
HEAD
LOCFLO
POSTM
PRINT
RPLATE
SCALEV
TRNSIT

METHOD OF SOLUTION

The surface integration logic routine CONIC, RPLATE, or CPLATE is called for surface geometry. Routine LOCFLO is called with the surface point, and local flow properties are obtained at this point. Then routines CONTIN and FREEM are called to obtain surface impact pressure and heating rates. With all the necessary surface information now obtained, the forces and torques are calculated and integrated over the body. These resultant forces and torques are then transformed into the exit system.

SUBROUTINE NAME: ITSUB

DESCRIPTION

This subroutine controls the iterative solution of any set of equations which can ultimately be expressed as a function of one variable. The routine can also be used to control an integration loop. It is used to solve for a real gas velocity in the impingement program.

CALLING SEQUENCE

CALL ITSUB (FOFX, X, SAVE, CONV, NTIMES)

(FOFX) - function of X which is driven to zero

(X) - variable which is iteratively solved for

(SAVE) - program control, i.e., SAVE(1) is control counter, SAVE(2) is X increment

(CONV) - convergence criteria

(NTIMES) - maximum number of iterations

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

ITSUB modifies (X) in the proper direction by the decrement value (SAVE(2)) until the root has been bracketed. The method of false position is then used to modify X until the solution is reached. Immediately after entering ITSUB each time, the function is inspected for convergence. If the function has converged, a program control is set, and computer control is transferred to the calling routine.

SUBROUTINE NAME: JPLGRD

DESCRIPTION

This subroutine changes the gas properties read by subroutine JPLTAP from MKS to English units. Also it calculates weighting factors to be used in the interpolation of temperature and pressure in subroutine TABLE.

CALLING SEQUENCE

```
CALL JPLGRD
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/TABCOM/
COMMON/TAPEFO/
COMMON/GASCON/
JPLTAP

METHOD OF SOLUTION

Not applicable

SUBROUTINE NAME: JPLTAP

DESCRIPTION

This subroutine reads the gas properties which were written by the Method-of-Characteristics program (Reference 4) on tape in binary form. This gas data originated from the NASA-Lewis Thermochemical Program (Reference 6). It stores the gas property data, in MKS units, as a function of entropy and velocity.

This subroutine also saves data on: 1. the number of entropy cuts; 2. the number of velocity cuts per entropy cut.

CALLING SEQUENCE

```
CALL JPLTAP
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/DUMPCO/
COMMON/TABCOM/
COMMON/TAPEFO/
UTILITY – none

METHOD OF SOLUTION

Not applicable

SUBROUTINE NAME: JWBTSS

DESCRIPTION

This subroutine performs all the flow field tape search operations for the trajectory program. The routine is entered with the coordinates of a point on the composite body, and a linear interpolation within the characteristics mesh produces flow field data at the desired point. The flow field has previously been written on tape, in binary form, by Lockheed's method-of-characteristics program.

CALLING SEQUENCE

CALL JWBTSS (POINT)

where

Point (1) = radial or vertical coordinate of desired point

Point (2) = axial coordinate of desired point

Point (3) = Mach number at desired point

Point (4) = flow angle at desired point

Point (5) = entropy level of flow at desired point

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/DUMPCO/
COMMON/FLAG/
JPLGRD
READB
READF
TAPMOV

METHOD OF SOLUTION

The main function of the routine is to locate the body point within the flow field or outside of its boundaries. If the body point is found to be within

the flow field, then flow field data is obtained at the location. This search operation is outlined in a series of steps performed by JWBTSS.

1. The routine first reads in the flow field boundary points and determines if it is possible for the body point to be inside the flow field. If this test is negative, a flag is set accordingly and control is returned to the calling routine LOCFLO.

   If the test is positive, step 2 is taken.

2. Characteristic lines are read in (one at a time) and retained in storage. As the lines are being read, each characteristic point is checked in an effort to bracket the body point. The search is conducted in a down-stream direction, however, if the new point is upstream of the previous point, the tape is backspaced six lines and bracketing is again attempted. If the tape is searched one hundred lines down stream of the starting point, or if the flow field boundary is reached, the tape is rewound and the search is started again.

3. If the body point is bracketed, four to eight of the surrounding characteristic points are used in a linear interpolation to determine Mach number, flow angle, and entropy.

Three flags are used as indicators of the results of the tape search.

1. IFLAG = 1 - indicates boundary has not been read in.
   = 2 - indicates boundary has been read in.
2. JFLAG = 1 - indicates last body point was not in flow field.
   = 2 - indicates last body point was in flow field
3. KFLAG = 1 - indicates current body point is not in flow field
   = 2 - indicates current body point is in flow field

SUBROUTINE NAME: LOCFLO

DESCRIPTION

Routine LOCFLO is the main control for determining the flow properties at a particular point.

CALLING SEQUENCE

CALL LOCFLO (NONUNI)

NONUNI = 0 If flow properties are desired at composite axis

= 1 If flow porperties are desired at subshape axis

= 2 If flow properties are desired at elemental areas

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/AMB/
COMMON/FLAG/
COMMON/GASFLO/
COMMON/GASCON/
COMMON/INPUT/
COMMON/LOCAL/
COMMON/ORIENT/
COMMON/POSVEC/
COMMON/QRCOM/
COMMON/SURFO/
COMMON/UNIT/
ADDV
JWBTSS
POFEM
POSTM
RGVOFM
RHOFEM
TABLE
TOFEM
VMAG

METHOD OF SOLUTION

The position of the body point is transformed from it's particular reference system into the exit plane (E) system. The degree of transformation required is dependent on the type body point, (1) composite system center of gravity requires no transformation, (2) subshape origins require transformation from the composite to the exit system, (3) surface points require transformation from the subshape system to the composite system and then to the exit system (see Reference 1 for mathematical explanation).

The three-dimensional body point thus obtained is converted into a two-dimensional plane and the tape search routine (JWBTSS) is called to acquire flow field data at this point.

From the flow field data obtained in JWBTSS (Mach number, flow angle, and Entropy) the desired properties (density, pressure, temperature, and velocity) are derived and stored in common. The scalar velocity is converted into a vector quantity and control is returned to the calling routine (IMPNGE).

SUBROUTINE NAME: ONEVAR

DESCRIPTION

This subroutine is used by subroutine PROPTY to provide interpolation between transport property data points as a function of temperature.

CALLING SEQUENCE

CALL ONEVAR (ARGUMT, NXDIR, TABLE, NOTAB, NX, OUTPUT)

where: ARGUMT is input interpolation argument (X)
NXDIR is type of interpolation, 1 is linear, 2 is quadratic
TABLE is set of X values followed by the Y values
NX is the variable number
NOTAB is table load capacity
OUTPUT is interpolated value of y=f(x)

UTILITY ROUTINES AND COMMON REFERENCES

COMMON - None

UTILITY - None

METHOD OF SOLUTION

This subroutine is an interpolation routine which solves for a function of one variable, i.e., y = f(x). The interpolation is performed using LaGrange's interpolation formula. The increments for the independent variable (x), may or may not be equidistant. The smaller the increment, the smaller the error will be in the answer, especially in areas of extreme curvature. The error involved is the error committed by replacing the given function by the polynominal and can be defined as

$$\text{ERROR} = f(x) - \phi(x)$$

also,

$$\text{ERROR} = \frac{f^{(n+1)}(\xi)}{(n+1!)} \; (x - x_0)\,(x - x_1) \ldots (x - x_n)$$

where ξ is some value of x between x_0 and x_n, and ϕ is the polynominal in x. The interpolation can be either linear or quadratic.

SUBROUTINE NAME: PLUME

DESCRIPTION

This subroutine provides the procedure for estimating heating rates at each point along each streamline. It supplies laminar, turbulent and transitional heating rates and a momentum Reynolds number transition criteria for choosing the proper flow regime.

CALLING SEQUENCE

```
CALL PLUME
```

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/FRACTN/
COMMON/HEATER/
PROPTY

METHOD OF SOLUTION

Local flow properties along a streamline are supplied to this subroutine by subroutine CONTH. These local properties and velocity gradients are used to provide a transform to the familiar flat plate equations for laminar and turbulent flow. The streamline divergence due to both body curvature and pressure gradients are estimated, and heating rates to the body are calculated. This procedure is followed along each streamline from the stagnation point to end of the body. Real gas transport properties are obtained using subroutine PROPTY.

FUNCTION NAME: POFEM

DESCRIPTION

This function computes the local static pressure as a function of Mach number and entropy.

CALLING SEQUENCE

P = POFEM (EM, S)

where (P) is the resultant static pressure found from the Mach number (EM) and entropy (S). NOTE: The appropriate values of the gas properties must be stored in common upon entry to this routine.

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
UTILITY - None

METHOD OF SOLUTION

Thermally perfect gas relationships are used to find the pressure.

$$p = p_o e^{-S/R} \left(1 + \frac{\gamma - 1}{2} M^2\right)^{-\frac{\gamma}{\gamma-1}}$$

SUBROUTINE NAME: POSTM

DESCRIPTION

This subroutine transforms a vector from one rectangular coordinate system to another by multiplying the vector by a transformation matrix.

CALLING SEQUENCE

CALL POSTM (VA, T, VB)

where VA is any vector, T is the 3 by 3 transformation matrix, and VB is the transformation of VA.

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

The components of the transformed vector (VB) are formed in the following way:

$$VB(1) = VA(1)* T(1,1) + VA(2)* T(2,1) + VA(3)* T(3,1)$$

$$VB(2) = VA(1)* T(1,2) + VA(2)* T(2,2) + VA(3)* T(3,2)$$

$$VB(3) = VA(1)* T(1,3) + VA(2)* T(2,3) + VA(3)* T(3,3).$$

SUBROUTINE NAME: PRINT

DESCRIPTION

This routine writes header information and body surface data at each elemental area.

CALLING SEQUENCE

CALL PRINT (IFLOW)

where

IFLOW = 0 - print header
IFLOW = 1 - data is for continuum regime
IFLOW = 2 - data is for freemolecular regime
IFLOW = 3 - data is for transitional regime

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASFLO/
COMMON/INPUT/
COMMON/LOCAL/
COMMON/PQOA/
COMMON/QRCOM/
COMMON/SURFO/
HEAD

METHOD OF SOLUTION

Not applicable

SUBROUTINE NAME: PROPTY

DESCRIPTION

This subroutine provides the calculational procedure for determining the local gas molecular weight, viscosity, thermal conductivity and specific heat. The subroutine is basically limited to gases containing mixtures of CO_2, H_2, H_2O, H, OH, CO, N_2 and HO_2; however, new data tables may be added when needed, and data currently exists for over 300 species.

CALLING SEQUENCE

CALL PROPTY (TLOCAL, VISCOS, COND, CP, OLWT)

where: TLOCAL is the gas temperature
VISCOS is the gas viscosity
COND is the gas thermal conductivity
CP is the gas specific heat
OLWT is the gas molecular weight

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/FRACTN/
ONEVAR

METHOD OF SOLUTION

The mole fractions of the individual gas species are inputs. The individual gas specie transport properties (viscosity and thermal conductivity) are then calculated through use of the Lenard-Jones potential model. The specific heat is calculated from a five polynominal curve fit, and the molecular weight of each species is a known quanity. The individual species property values are then averaged by the subroutine to give values for the mixture transport proerties.

SUBROUTINE NAME: READ

DESCRIPTION

This routine reads and writes the input data, calculates initial Eulerian angles of the composite system, and produces the transformation matrices [TIC] and [TCI] (see Reference 1).

CALLING SEQUENCE

CALL READ

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/AMB/	CROSS
COMMON/FORCFO/	HEAD
COMMON/INPUT/	TRANS
COMMON/LOAD/	TRAJ
COMMON/LOCAL/	
COMMON/MASSES/	
COMMON/ORIENT/	
COMMON/POSVEC/	
COMMON/STAGCO/	
COMMON/SURFO/	
COMMON/TIMEFO/	
COMMON/UNIT/	
COMMON/UNITV/	
COMMON/FLAG/	

METHOD OF SOLUTION

The unit vectors of the composite system are formed. The initial Eulerian angles are then obtained from functions of these unit vectors. Unit vectors for each of the subshape systems are also formed and the invariant transformation matrixes from the composite system to the subshape systems

are derived from them (see Reference 1 for a more complete mathematical explanation). All other input quantities are merely read in, converted to working units when necessary, stored into common, and printed out.

SUBROUTINE NAME: READB

DESCRIPTION

Subroutine READB reads the position of the boundary points of the flow field.

This information is read from a binary tape produced by the method of characteristics program.

Subroutine JWBTSS uses this to determine boundary position of flow field.

CALLING SEQUENCE

CALL READB (X, R, ITOT1)

X = X coordinate of last point on characteristic line

R = R coordinate of last point on characteristic line

ITOT1 = Number of points on the line

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

Not applicable

SUBROUTINE NAME: READF

DESCRIPTION

SUBROUTINE READF reads one characteristic line from flow field tape and saves the following data at each point:

1. X position of the point,
2. R position of the point,
3. Mach number at the point,
4. Flow angle at the point,
5. Entropy at the point.

This information is read from a binary tape produced by the method of characteristics program.

CALLING SEQUENCE

CALL READF (J, ITOT1)

J — number of characteristic line

ITOT1 — number of points in characteristic line

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/DUMPCO/

UTILITY - None

METHOD OF SOLUTION

Not applicable

FUNCTION NAME: RGVOFM

DESCRIPTION

This subroutine finds velocity as a function of Mach number and entropy. The difference between this routine and VOFEM is that the gas properties are not known prior to entry.

CALLING SEQUENCE

V = RGVOFM (S, EM)

where (V) is the resultant velocity formed from entropy (S) and Mach number (EM).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/TAPEFO/
COMMON/TABCOM/
TABLE
VOFEM
EMOFV
ITSUB

METHOD OF SOLUTION

The real gas tables have, as independent variables, entropy and velocity. If the velocity is not known, an iterative solution must be employed to find the velocity from Mach number and entropy.

FUNCTION NAME: RHOFEM

DESCRIPTION

This function computes the local density as a function of Mach number and entropy.

CALLING SEQUENCE

RHO = RHOFEM (EM, S)

where RHO is the resultant density found from local Mach number and local entropy. NOTE: The appropriate values of the gas properties must be stored in common upon entry to this routine (see Subroutine Table).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
POFEM

METHOD OF SOLUTION

Thermally perfect gas relationships are used to find the density.

$$\rho = \rho_o^* \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{-1}{\gamma-1}}$$

SUBROUTINE NAME: RPLATE

DESCRIPTION

This routine divides a rectangular plate into elemental segments and provides the necessary logic to perform a surface integration.

CALLING SEQUENCE

CALL RPLATE

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/LOCAL/
COMMON/SURFO/
UTILITY ROUTINES-None

METHOD OF SOLUTION

The plate is divided into a specified number of rectangles. The area, unit normal and position vector of each rectangle is also calculated.

SUBROUTINE NAME: SCALEV

DESCRIPTION

This subroutine changes the magnitude of a vector.

CALLING SEQUENCE

CALL SCALEV (VA, F, VB)

where VA is any vector to be scaled, F is the scalar, and VB is the new vector.

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

The components of vector VA are multiplied by the scalar F to form the vector VB. VB is returned.

$$VB = F\ (VA)$$

SUBROUTINE NAME: TABLE

DESCRIPTION

This subroutine utilizes real or ideal gas information obtained from the flow field tape to calculate properties locally in the flow. The maximum size of the array used by (TABLE) is limited to five gas properties (V, R, γ, T_o, P_o) at thirteen velocity "cuts" for each of nine entropy cuts.

CALLING SEQUENCE

CALL TABLE (SS, VV)

where (SS) is the local entropy and (VV) is the local velocity at the point of interest.

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/TAPEFO/
COMMON/TABCOM/
COMMON/GASCON/
TOFV
POFEM
EMOFV

METHOD OF SOLUTION

The routine is entered with the local entropy (SS) and velocity (VV). A test is then made to determine if the gas is real or ideal. If the test indicates an ideal gas, the local properties are set to those stored in the (TAB) common array. If the test indicates real gas, a double interpolation scheme is utilized to locate gas properties between tabulated values of velocity and entropy. In the case of an entry velocity beyond the range of the table, an ideal gas extrapolation from the last table value is made to locate the gas properties.

SUBROUTINE NAME: TAPMOV

DESCRIPTION:

This subroutine moves the binary flow field tape through the gas property data to the start of the flow field information.

CALLING SEQUENCE

CALL TAPMOV

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

Not applicable

FUNCTION NAME: TOFEM

DESCRIPTION

This function computes the local static temperature as a function of Mach number. The gas properties at the point of interest are known prior to entry. TOFEM and TOFV are quite similar; the difference being if Mach number or velocity is the known quantity.

CALLING SEQUENCE

T = TOFEM (EM)

where (T) is the one-dimensionally calculated local static pressure which exists at the Mach number (EM).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/

METHOD OF SOLUTION

The calorically perfect gas relationship

$$T = \frac{T_o}{1 + \frac{\gamma - 1}{2} M^2}$$

is solved for static temperature at the local Mach number.

FUNCTION NAME: TOFV

DESCRIPTION

This function computes the local static temperature as a function of velocity. The gas properties at the point of interest are known prior to entry. TOFV and TOFEM are quite similar; the difference being if Mach number or velocity is the known variable.

CALLING SEQUENCE

$$T = TOFV(V)$$

where (T) is the one-dimensionally calculated local static pressure which exists at the velocity (V).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/

UTILITY - None

METHOD OF SOLUTION

The calorically perfect gas relationship

$$T = T_o - \frac{V^2}{2R}\left(\frac{\gamma - 1}{\gamma}\right)$$

is solved for static temperature at the local velocity.

SUBROUTINE NAME: TRANS

DESCRIPTION

This subroutine transposes a matrix and returns the transposed matrix.

CALLING SEQUENCE

CALL TRANS (T1, T2)

where T 1 is the input matrix and T 2 is the transposed matrix

UTILITY ROUTINES AND COMMON REFERENCES

NONE

METHOD OF SOLUTION

The rows and columns are interchanged in T 1 to form T 2.

$$T1 = \begin{bmatrix} T_{11} & T_{12} & T_{13} \\ T_{21} & T_{22} & T_{23} \\ T_{31} & T_{32} & T_{33} \end{bmatrix} \text{ and } T2 = \begin{bmatrix} T_{11} & T_{21} & T_{31} \\ T_{12} & T_{22} & T_{32} \\ T_{13} & T_{23} & T_{33} \end{bmatrix}$$

SUBROUTINE NAME: TRNSIT

DESCRIPTION

The routine determines transitional values for heating rates and impact pressure using an empirical function of the Knudsen number (see Reference 1)

CALLING SEQUENCE

CALL TRANSIT (XCONT, XFREEM, AKN, TRANSF)

XCONT - continuum value of heating rate or impact pressure
XFREEM - free molecular value of heating rate or impact pressure
AKN - Knudsen number
TRANSF - returned transitional value of heating rate or impact pressure

UTILITY ROUTINES AND COMMON REFERENCES

NONE

METHOD OF SOLUTION

Transitional values are obtained from the following relationship:

$$\text{TRANSF} = \text{XCONT} + (\text{XFREEM} - \text{XCONT}) \left\{ \sin^2 \left[\pi \left(1/3 + 1/6 \log_{10} \text{AKN} \right) \right] \right\} .$$

FUNCTION NAME: UOFEM

DESCRIPTION

This function computes the Mach angle at a local Mach number.

CALLING SEQUENCE

EMU = UOFEM (EM)

where (EMU) is the Mach angle which exists at the local Mach number (EM).

UTILITY ROUTINES AND COMMON REFERENCES

None

METHOD OF SOLUTION

The following equation

$$\mu = \tan^{-1}\left(\frac{1}{\sqrt{M^2 - 1}}\right)$$

is solved for the local Mach angle.

FUNCTION NAME: UOFV

DESCRIPTION

This function computes the Mach angle at a local velocity.

CALLING SEQUENCE

EMU = UOFV (V)

where (EMU) is the Mach angle which exists at the local velocity (V).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON - None

UOFEM

EMOFV

METHOD OF SOLUTION

The local velocity is converted into a Mach number using (EMOFV). Function (UOFEM) is then entered with the calculated Mach number and the Mach angle obtained from the following equation.

$$\mu = \tan^{-1}\left(\frac{1}{\sqrt{M^2 - 1}}\right)$$

FUNCTION NAME: VMAG

DESCRIPTION

This function determines the magnitude of a vector.

CALLING SEQUENCE

FUNCTION = VMAG (VA)

where VA is any vector.

UTILITY ROUTINES AND COMMON REFERENCES

NONE

METHOD OF SOLUTION

$$VMAG = \sqrt{VA(1)^2 + VA(2)^2 + VA(3)^2}$$

FUNCTION NAME: VOFEM

DESCRIPTION

This function computes velocity as a function of Mach number. Ideal gas relations are used and the gas properties are known prior to entry.

CALLING SEQUENCE

V = VOFEM (EM)

where (V) is the local velocity which corresponds to the local Mach number (EM).

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
TOFEM

METHOD OF SOLUTION

The ideal gas relationship

$$V = \sqrt{\frac{R\gamma(T_o - T)}{\left(\frac{\gamma - 1}{2}\right)}}$$

is solved for velocity. Local static temperature (T) is obtained from the input Mach number.

SUBROUTINE NAME: WEAK

DESCRIPTION

This subroutine determines the independent variables (SD, VD) downstream of a weak oblique shock. The gas properties upstream of the shock are known prior to entry.

CALLING SEQUENCE

CALL WEAK (SU, VU, ERS, DELTA, SD, VD)

where (SU, VU) are the upstream entropy and velocity, (EPS, DELTA) are the shock angle and turning angle, and (SD, VD) are the downstream entropy and velocity.

UTILITY ROUTINES AND COMMON REFERENCES

COMMON/GASCON/
TABLE
EMOFV
POFEM
RHOFEM
ENTROP
DELTAF

METHOD OF SOLUTION

From the known upstream entropy and velocity, the local gas properties, pressure, density, and upstream Mach number are calculated. The entropy rise across the shock is added to the upstream entropy to get total downstream entropy. Downstream velocity is calculated from the following relationship.

$$V_D = \frac{V_u \cos(\xi)}{\cos(\xi - \delta)}$$

Section 3

INPUT INSTRUCTIONS

INPUT INSTRUCTIONS

Card 1

This card contains a title or heading which is used for run identification.

Format	Column	Item
12A6	1-72	Title or Heading

Card 2

This card contains input options necessary to control program execution.

Format	Column/Option	Item
16I5	5/IOPT(1)	Body Geometry
		0 - Composite Body Nonaxisymmetric
		1 - Composite Body Axisymmetric
	10/IOPT(2)	Desired Heating Options
		0 - Calc. Continuum and Free Molecular Heating
		1 - Calc. Continuum Heating Only
		2 - Calc. Free Molecular Heating Only
	15/IOPT(3)	Uniform Flow Assumptions
		0 - Composite Vehicle in Uniform Flow
		1 - Subshapes in Uniform Flow
		2 - Elemental rings in uniform flow at maximum impingement
		3 - Elemental areas in uniform flow
	18-20/IOPT(4)	Number of Subshapes in Composite Body
		N - Number of Subshapes (100 max)

Format	Column/Option	Item
16I5	25/IOPT(5)	Type Output Desired 0 - Full Output (Data at Each Elemental Area) 1 - Force and Torque Summary Only
	30/IOPT(6)	Flow Field Type 1 - Flow Field is Axisymmetric 0 - Flow Field is Two-Dimensional
	35/IOPT(7)	(-NFRACT) - number of mole fractions for gas used
	40/IOPT(8)	(-ISTAG) - number of subshape where stagnation point located
	45/IOPT(9)	Type Local Properties 1 - Newtonian assumption 2 - Modified Newtonian 3 - Isentropic expansion
	46-50/IOPT(10)	Surface Integration Control N - Number of X Increments for Conic Subshape or, Number of R Increments for Circular Plate Subshape or, Number of Y Increments for Rectangular Plate Subshape
	51-55/IOPT(11)	Surface Integration Control N - Number of Angular (PHI) Increments for Conic and Circular Plate Subshape or, Number of Z Increments for Rectangular Plate Subshape
	56-60/IOPT(12)	Case Number
	65/IOPT(13)	0 - Impingement Program Dimensioned in Inches 1 - Impingement Program Dimensioned in Feet
	70/IOPT(14)	0 - Flow Field Dimensioned in Inches 1 - Flow Field Dimensioned in Feet

Format	Column/Option	Item
16I5 (cont)	75/IOPT(15)	Local Property Printout
		0 - No Printout 1 - Pring Local Properties
	80/IOPT(16)	Not Presently Used

Card 3

This card contains initial values for the position vector (REC). This vector describes the location of the composite body ("C") system with respect to the exit ("E") system. The vector is measured in the exit ("E") reference plane. The position vector (REC) is measured in inches or feet.

Format	Column	Item
6E10.6		X, Y, Z Components of Position
	1-10	REC(X) (in. or ft.)
	11-20	REC(Y) (in. or ft.)
	21-30	REC(Z) (in. or ft.)

Card 4

This card contains initial values for the composite system unit vectors (IC and JC). For an axisymmetric body, only the unit vector in the X direction (IC) need be input. The unit vectors are measured in the exit ("E") reference system.

Format	Column	Item
6E10.6		X, Y, Z Components of IC
	1-10	IC(X)
	11-20	IC(Y)
	21-30	IC(Z)
		X, Y, Z Components of JC
	31-40	JC(X)
	41-50	JC(Y)
	51-60	JC(Z)

Note: JC values are not needed for an axisymmetric composite shape.

Card 5

This card reads the mole fractions of the flow field gas composition. There are presently eight of the most common constituients being considered and they must be input in the correct order. The sum of the mole fraction must equal one (1).

Format	Column	Item
8E10.6	1-10	CO_2 mole fraction
	11-20	H_2 mole fraction
	21-30	H_2O mole fraction
	31-40	H_1 mole fraction
	41-50	OH mole fraction
	51-60	CO mole fraction
	61-70	N_2 mole fraction
	71-80	O_2 mole fraction

Cards 6A, 6B, 6C, 6D, 6E, 6F

These cards contain constants used in the pressure and heating routines.

6A

Format	Column	Item
3E10.6	1-10	(REFL) - Reference Length (in. or ft.)
	11-20	(REFD) - Reference Diameter (in. or ft.)
	21-30	(DIA) - Molecular Diameter (in. or ft.)

6B

Format	Column	Item
3E10.6	1-10	(HO) - Total Enthalpy (Btu/lbm)
	11-20	(TW) - Wall Temperature (^{o}R)
	21-30	(K) - Thermal Conductivity (Btu/sec^{o}R ft.)

6C

3E10.6	1-10	(CP) - Specific Heat at Constant Pressure (Btu/lbm^{o}R)
	11-20	(EMU) - Viscosity (lbf sec/ft^{2})
	21-30	(RE) - Reynolds Number (Nondimensional)

6D

3E10.6	1-10	(HCHEM) - Total Chemical Enthalpy (Btu/lbm)
	11-20	(XTE) - Transfer Distance from Flow Field (in. or ft.) Nozzle Throat to Exit Plane (in. or ft.)
	21-30	(AB) - Characteristic Area for Outside of Plume Drag Calculations (in.^{2} or ft.^{2})

6E

3E10.6	1-10	(CD) - Drag Coefficient for Outside of Plume Calculation
	11-20	(PAMB) - Pressure Outside Plume (PSF)
	21-30	(TAMB) - Temperature Outside Plume (^{o}R)

6F

Format	Column	Item
3E10.6	1-10	(CPMAX) - Maximum Pressure Coefficient
	11-20	(XSTAG) - Axial Distance from Subshape origin to Stagnation Point
	21-30	(DELPHI) - Constant Angular Increment off Stagnation Line (used when IOPT(3) equals 2)

Cards 7N

These cards identify the type surface (conic, retangular plate, circular plate) and the coefficients of it's surface equation. There will be as many of these cards as there are subshapes, up to a maximum of 100.

Format	Column	Item		
II, 9X, 7E10.6		Conic	R Plate	C Plate
	1	1	2	3
	5	K		
	11-20	A	Y MAX (in. or ft.)	ROUTER (in. or ft.)
	21-30	B	Z MAX (in. or ft.)	RINNER (in. or ft.)
	31-40	C		
	41-50	D		
	51-60	E		
	61-70	XNOSE (in. or ft.)		
	71-80	XBASE (in. or ft.)		

K = 1 for other then cylinder or cone
= 2 for cylinder
= 3 for cone

The conic equation is of the form

$$R = A\left(\sqrt{B + CX + DX^2} + E\right),\ XNOSE \le X \le XBASE$$

And DA = R Δ R Δ PHI.

For the circular plate

$DA = R \Delta R \Delta PHI \qquad RINNER \leq R \leq ROUTER$

the rectangular plate is defined as

$DA = \Delta Y \Delta Z, \quad O \leq Y \leq Y\,MAX$

$O \leq Z \leq Z\,MAX$

Cards 8N

These cards contain the position vectors (RCI) from the composite structure to the subshape origins. RCI is measured in the composite system reference frame. There will be as many of these cards as there are subshapes, up to a maximum of 100.

Format	Column	Item
3E10.6	1-10	RCI(X) - X Component of Position Vector (in. or ft.)
	11-20	RCI(Y) - Y Component of Position Vector (in. or ft.)
	21-30	RCI(Z) - Z Component of Position Vector (in. or ft.)

NOTE: RCI is measured in the composite ("C") system and "points" to the subshape origins.

Cards 9N

These cards contain the subshape unit vectors (II and JI). If the subshape is of the rectangular plate type, both II and JI must be specified, otherwise only II is needed. There will be one of these cards for each subshape up to a maximum of 100. IT and JI are measured in the composite reference frame.

Format	Column	Item
6E10.6	1-10	II(X), X Component of Unit Vector I
	11-20	II(Y), Y Component of Unit Vector I
	21-30	II(Z), Z Component of Unit Vector I
	31-40	JI(X), X Component of Unit Vector J
	41-50	JI(Y), Y Component of Unit Vector J
	51-60	JI(Z), Z Component of Unit Vector J

Section 4

EXAMPLE PROBLEM

EXAMPLE PROBLEM

I. COMPOSITE BODY

The body is a combination of cylindrical subshapes and truncated cone subshapes. The body is designed to simulate a model of the J-2 engine thrust structure and nozzle. See Figure 1 for a graphic representation of the body geometry.

II. BODY ENVIRONMENT

The composite body is immersed in a nitrogen plume generated by a model of the J-2 engine O_2/H_2 burner. This plume has been calculated by Lockheed's Method of Characteristics Program (References 2, 3 and 4) and stored on magnetic tape. The centerline of the model J-2 exit plane is located relative to the model O_2/H_2 burner exit plane by the following coordinates;

1. Axial Displacement = 13.519 inches
2. Vertical Displacement = 9.3877 inches
3. No displacement out of the X, Y Plane (see Figure 1 for a graphic representation of the composite body location).

III. GENERAL INFORMATION NECESSARY TO SET UP PROBLEM

Body Reference Length = 15.03 inches

Body Reference Diameter = 7.87 inches

Average Molecular Diameter = 10^{-7} inches

Total Enthalpy of Plume = 500 Btu/lbm$^{\circ}$R

Wall Temperature = 500°R

Distance from 3rd Subshape to Stagnation Point = 0.5 inches

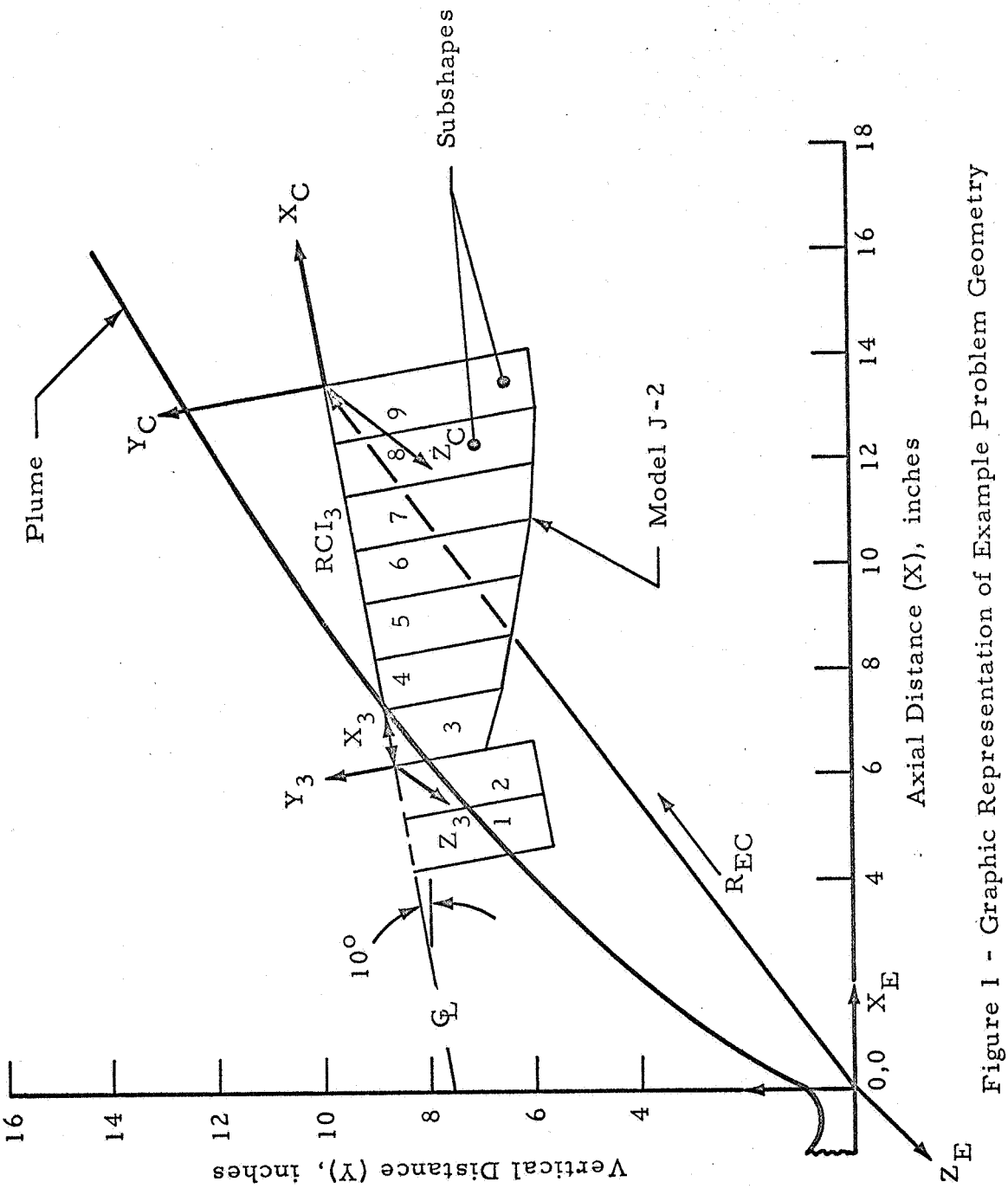


Figure 1 - Graphic Representation of Example Problem Geometry

IV. DESIRED RESULTS

Heating rates and impact pressure along the line of maximum impingement (stagnation line) are of primary interest.

V. EXAMPLE PROGRAM INPUT INSTRUCTIONS

Card 1

Any run title up to 72 words may be used. "Douglas Comparison of O_2/H_2 Burner Impinging on J-2" was used for this case.

Card 2

This card contains program control options

IOPT(1) = 1 - composite body is axisymmetric

IOPT(2) = 0 - continuum and free molecular heating is desired

IOPT(3) = 2 - assume each elemental ring in uniform flow at maximum impingement point

IOPT(4) = 9 - there are nine (9) total subshapes in the composite body

IOPT(5) = 0 - full output desired

IOPT(6) = 1 - flow field is axisymmetric

IOPT(7) = 8 - there are presently eight (8) possible molecular species available

IOPT(8) = 3 - the stagnation point is somewhere on the third subshape

IOPT(9) = 1 - Newtonian flow is assumed

IOPT(10) = 1 - only one (1) ring per subshape

IOPT(11) = 12 - allow twelve (12) equal angular increments for each ring

IOPT(12) = 1 - case number (1)

IOPT(13) = 0 - impingement program units of length are in inches

IOPT(14) = 0 - flow field program units of length are in inches

IOPT(15) = 0 - local property printout not required

IOPT(16) = 0 - not used

Card 3

The composite body reference system may be chosen anywhere on the body. For this case, the centerline of the model J-2 exit plane was chosen. Components of the position vector REC are from the centerline of the model O_2/H_2 burner exit plane to the composite body origin. The vector "points" from the exit "E" system to the composite "C" system. (See Figure).

REC (X) = 13.519 inches

REC (Y) = 9.8877 inches

REC (Z) = 0 inches

Card 4

The centerline of the model J-2 engine is inclined 10^{o} with respect to the model O_2/H_2 burner centerline; therefore, there will be X_E and Y_E components in the unit vector I_C.

IC (X) = .98481

IC (Y) = .17365

IC (Z) = 0.

Since the model J-2 is an axisymmetric body it is not necessary to input J_C, the program will calculate it and print it out for inspection.

Card 5

The model O_2/H_2 burner plume has been simulated by heated N_2; therefore a mole fraction of 1.0 is input in the column for N_2 (Col. 61-70).

Card 6

6A

REFL = 15.030 inches - the axial length of the model J-2 engine

REFD = 7.87 inches - the model J-2 exit diameter

DIA = 10^{-7} inches - arbitrarily chosen to assure continuum flow

6B

H_O = 500 Btu/lbm$^{\circ}$R - total enthalpy of plume

TW = 500°R - wall temperature

K = 0 - calculated by program

6C

C_p = 0 - calculated by program

EMU = 0 - calculated by program

RE = 0 - calculated by program

6D

HCHEM = 0 - no chemical enthalpy

XTE = 0 - origin of O_2/H_2 plume already at exit plane

AB = 0 - not required for this calculation

6E

CD = 0 - not required for this calculation

PAMB = 0 - medium outside plume is a vacuum

TAMB = 0 - medium outside plume is a vacuum

6F

CPMAX = 0 - not required unless IOPT(9) = 2

XSTAG = .5 inches - distance from origin of 3rd subshape to stagnation point

DELPHI = 0 - 1st angular calculation desired at maximum impingement point.

Card 7N

The composite body consists of nine (9) subshapes. The first two are cylindrical, simulating part of the J-2 thrust structure, and the last seven (7) are truncated cones, simulating the model J-2 nozzle.

All nine equations are of the form

$$R_i = D_i X + E_i; \quad X_{nose} \leq x \leq X_{base}$$

and are represented by the type 1 (CONIC) equation; therefore, a (1) appears in Col. 1 for all nine.

The two cylindrical subshapes are 2.5937 inches in radius and are 1.0 inches long, therefore;

$A_1 = 1.0$ $A_2 = 1.0$

$B_1 = 0$ $B_2 = 0$

$C_1 = 0$ $C_2 = 0$

$D_1 = 0$ $D_2 = 0$

$E_1 = 2.5937$ $E_2 = 2.5937$

$X_{N_1} = 0$ $X_{N_2} = 0$

$X_{B_1} = 1.0$ $X_{B_2} = 1.0$

The first of the next seven truncated cones is 1.35 inches long and the last six are 1.0 inches in length; therefore,

$A_3 = 1.0$	$A_4 = 1.0$	$A_5 = 1.0$	$A_6 = 1.0$	$A_7 = 1.0$
$B_3 = 0$	$B_4 = 0$	$B_5 = 0$	$B_6 = 0$	$B_7 = 0$
$C_3 = 0$	$C_4 = 0$	$C_5 = 0$	$C_6 = 0$	$C_7 = 0$
$D_3 = .1936$	$D_4 = .16$	$D_5 = .16$	$D_6 = .1089$	$D_7 = .0729$
$E_3 = 1.76$	$E_4 = 2.2$	$E_5 = 2.6$	$E_6 = 3.0$	$E_7 = 3.33$
$X_{N_3} = -.35$	$X_{N_4} = 0$	$X_{N_5} = 0$	$X_{N_6} = 0$	$X_{N_7} = 0$
$X_{B_3} = 1.0$	$X_{B_4} = 1.0$	$X_{B_5} = 1.0$	$X_{B_6} = 1.0$	$X_{N_7} = 1.0$

$A_8 = 1.0$ $A_9 = 1.0$

$B_8 = 0$ $B_9 = 0$

$C_8 = 0$ $C_9 = 0$

$D_8 = .04$ $D_9 = .0169$

$E_8 = 3.6$ $E_9 = 3.8$

$X_{N_8} = 0$ $X_{N_9} = 0$

$X_{B_8} = 1.0$ $X_{B_9} = 1.0$

Card 8N

All the subshape origins lie on the composite system axis and are only displaced some distance RCI(X). The magnitude RCI(X) indicates the absolute distance from the composite system to each subshpae system. For this case, all subshapes are "upstream" of the composite system axis and therefore in a negative X direction (positive x in the composite system is in the direction of flow)

$RCI(X)_1 = -9.35$ in. $RCI(Y)_1 = 0$ $RCI(Z)_1 = 0$

$RCI(X)_2 = -8.35$ in. $RCI(Y)_2 = 0$ $RCI(Z)_2 = 0$

$RCI(X)_3 = -7.0$ in. $RCI(Y)_3 = 0$ $RCI(Z)_3 = 0$

$RCI(X)_4 = -6.0$ in. $RCI(Y)_4 = 0$ $RCI(Z)_4 = 0$

$RCI(X)_5 = -5.0$ in. $RCI(Y)_5 = 0$ $RCI(Z)_5 = 0$

$RCI(X)_6 = -4.0$ in. $RCI(Y)_6 = 0$ $RCI(Z)_6 = 0$

$RCI(X)_7 = -3.0$ in. $RCI(Y)_7 = 0$ $RCI(Z)_7 = 0$

$RCI(X)_8$ = -2.0 in. $RCI(Y)_8 = 0$ $RCI(Z)_8 = 0$

$RCI(X)_9$ = -1.0 in. $RCI(Y)_9 = 0$ $RCI(Z)_9 = 0$

Card 9N

All subshape origins are co-incident with the composite system origin and all subshapes are axisymmetric; therefore, only the X component appears in each unit vector (II) and only (II) need by input

$I_1(X) = 1.0$ $I_1(Y) = 0$ $I_1(Z) = 0$

$I_2(X) = 1.0$ $I_2(Y) = 0$ $I_2(Z) = 0$

$I_3(X) = 1.0$ $I_3(Y) = 0$ $I_3(Z) = 0$

$I_4(X) = 1.0$ $I_4(Y) = 0$ $I_4(Z) = 0$

$I_5(X) = 1.0$ $I_5(Y) = 0$ $I_5(Z) = 0$

$I_6(X) = 1.0$ $I_6(Y) = 0$ $I_6(Z) = 0$

$I_7(X) = 1.0$ $I_7(Y) = 0$ $I_7(Z) = 0$

$I_8(X) = 1.0$ $I_8(Y) = 0$ $I_8(Z) = 0$

$I_9(X) = 1.0$ $I_9(Y) = 0$ $I_9(Z) = 0$

This completes the example problem input. Results of the computer run described here have been converted into graphical form and are compared to test data in Reference 1.

REFERENCES

1. Ratliff, A. W., "Analysis of Heating Rates and Forces on Bodies Subject to Rocket Exhaust Plume Impingement," HREC-1150-1, LMSC/HREC A791230, Lockheed Missiles & Space Company, Huntsville, Ala., March 1968.

2. Prozan, R. J., "Development of a Method of Characteristics Solution for Supersonic Flow of an Ideal, Frozen or Equilibrium Reacting Gas Mixture," LMSC/HREC A783535, Lockheed Missiles & Space Company, Huntsville, Ala., April 1966.

3. Ratliff, A. W., "Comparisons of Experimental Supersonic Flow Fields with Results Obtained by Using a Method of Characteristics Solution," LMSC/HREC A782592, Lockheed Missiles & Space Company, Huntsville, Ala., April 1966.

4. Butler, H. W., "Description of a Digital Computer Program for Nozzle and Plume Analysis by the Method of Characteristics," LMSC/HREC A782592, Lockheed Missiles & Space Company, Huntsville, Ala., December 1966.

5. Sentman, Lee H., "Free Molecule Flow Theory and Its Application to the Determination of Aerodynamic Forces," LMSC-448514, Lockheed Missiles & Space Company, Sunnyvale, Calif., 1 October 1967.

6. Zeleznik, F. J. and S. Gordon, "A General IBM 704 or 7090 Computer Program for Computation of Chemical Equilibrium Compositions, Rocket Performance, and Chapman-Jouget Detonations," NASA TN D-1454, Lewis Research Center, Cleveland, Ohio, October 1962.