NASA TM - 81856

NASA Technical Memorandum 81856

NASA-TM-81856 19810019011

COMPUTER PROGRAM FOR NONLINEAR STATIC STRESS ANALYSIS OF SHUTTLE THERMAL PROTECTION SYSTEM - USER'S MANUAL

Gary L. Giles and
Maria Vallas

May 1981

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665

CONTENTS

SUMMARY

User documentation is presented for a computer program which considers the nonlinear properties of the strain isolator pad (SIP) in the static stress analysis of the shuttle thermal protection system (TPS). This program is generalized to handle an arbitrary SIP footprint including cutouts for instrumentation and filler bar. Multiple SIP surfaces can be defined to model tiles in unique locations such as leading edges, intersections, and penetrations. The nonlinearity of the SIP is characterized by experimental stress-displacement data for both normal and shear behavior. Stresses in the SIP are calculated using a Newton iteration procedure to determine the six rigid body displacements of the tile which develop reaction forces in the SIP to equilibrate the externally applied loads. This user documentation gives an overview of the analysis capabilities, a detailed description of required input data and an example to illustrate use of the program.

INTRODUCTION

The metal primary structure on the external surface of the space shuttle orbiter is protected from the thermal environment by an array of RSI (reusable surface insulation) tiles. These tiles are not attached directly to the shuttle surface or substrate but are mounted on an intermediate strain isolator pad as shown in figure 1(a). This pad prevents strain in the shuttle structure produced by thermal or aerodynamic loading from cracking the brittle tile material. In the baseline TPS configuration, the tiles have uniform properties through the thickness (undensified tiles). During TPS tests with undensified tiles under externally applied loads, failure was found to occur in the tile adjacent to the plane where it was bonded to the SIP. Therefore in this case, the failure stress can be taken as the through-the-thickness or normal stress in the SIP. As indicated in reference 1, the SIP has nonlinear stress-displacement properties.

The present report contains user documentation for a computer program which considers these nonlinear properties in a static analysis for calculating stresses in the SIP. The tile attached to the outside of the SIP is considered rigid and can have an arbitrary shape built up with block elements as shown in figure 1(b). This program is generalized to handle an arbitrary SIP footprint including cutouts for instrumentation (fig. 1(c)). Also, filler bar material which is usually located under the tile around the perimeter of the SIP but not attached to the tile can be considered. Multiple SIP and filler bar areas can be defined to model tiles with nonplanar surfaces in unique locations such as leading edges, intersections, and penetrations.

A brief description of the procedure used to calculate the SIP stresses is presented in the next section. A detailed description of the required input data and an example to illustrate the input and corresponding calculated results are given in the remainder of this report.

ANALYSIS PROCEDURE

The analysis procedure used in the computer program is a generalization and extension of the basic method described in reference 2. The SIP material has nonlinear stress-displacement relationships in both the through-the-thickness or normal and shear directions. These nonlinear curves are obtained from material tests and are represented in the program as tables of stress-displacement points. The tile is assumed to be a rigid body with six degrees of freedom (three translations and three rotations) at its center of gravity. External loads are applied to the tile as concentrated forces, pressures, and/or statically equivalent accelerations (g-load). The substrate under the SIP can be given a prescribed shape to represent mismatch from the manufacturing process, warpage of the tile, and/or deformations of the external surface of the structure such as those caused by buckling.

A Newton iteration procedure is used to calculate the displacements and rotations of the rigid tile for which the reaction forces from stresses in the SIP and filler bar material are in static equilibrium with the applied loads. The iteration is started using an assumed set of tile displacements and rotations. This position of the tile relative to the substrate is then used to calculate normal and shear displacements of the SIP material. The corresponding SIP stresses are obtained from the nonlinear stress-displacement curves. Stresses over each SIP surface are numerically integrated and the resulting forces and moments are summed at the center of gravity of the tile. The SIP reactions are compared to the external loads and the unbalanced or residual forces are calculated. During the numerical integration process, a Jacobian matrix is formed whose terms are the derivatives or changes in forces and moments produced by changes in the tile displacements and rotations. This Jacobian matrix and set of unbalanced forces is used to calculate an updated set of displacements and rotations by Newton's method. The iteration is continued until the SIP forces and external forces are equilibrated. Stresses in the SIP are then obtained from the stress-displacement material curves for the resulting values of local displacements over each SIP surface.

To provide capability to analyze all tile configurations the SIP footprints are defined with an arbitrary boundary of linear segments including cutouts for instrumentation or other penetrations. A general numerical integration method is provided to accommodate SIP areas with complex boundaries. In this method, the SIP and/or filler bar surfaces are divided into triangular regions. Within each region, the integration process assumes a linear distribution of the integrand over subtriangles which are defined using area coordinates as discussed in reference 3.

DESCRIPTION OF COMPUTER PROGRAM

The nonlinear SIP stress analysis program is operational on both the CDC NOS 1.3 and the CDC NOS/BE operating systems. The program is organized as a primary overlay (0,0) and two secondary overlays (1,0) and (2,0). The primary (0,0) and the (1,0) secondary overlays are used for input and generation of required geometry and applied load information. The majority of the subroutines in these two overlays were written by Al Dobrowski at Rockwell International for use in a linear SIP stress analysis program, reference 4. These subroutines were adapted with slight extensions and modifications where necessary for use in the nonlinear SIP analysis program. Overlay (2,0) contains the NASA-LaRC developed subroutines for implementing the nonlinear analysis procedure described in the previous section. The analysis program uses the SPAR data handling utilities, reference 5, to access the experimental stress-displacement curves which are maintained on a separate data file. The program generates a complete file of calculated data on TAPE6 which is usually output on a high speed printer. Additionally, a synopsis of stress distributions over each SIP pad is output on TAPE1 for viewing on an interactive terminal upon completion of the analysis run. A single line containing the maximum stress value from each analysis with some identification data is output on TAPE2 to form a table for runs with multiple tiles and/or multiple load conditions.

SUMMARY OF COMPUTER PROGRAM INPUT

The input data to the program is generally in list directed input form on 80-column card images as described in reference 6. In list directed form, the input data are free-field (can be located anywhere on the card image) and values are separated by one or more blanks, or by a comma or a slash, either of which may be preceded or followed by any number of blanks. The input deck contains commands beginning with a special designator word to indicate which of the various types of input data will follow. These commands are described herein in the general order that they would occur in an input deck.

Three commands are available for providing descriptive data to the program. These commands are:

"TITLE"

"CASEID"

"COMMENT"

Geometry input data are designated by the next group of commands.

Commands used to define node points are:

"NODE"

"CORNER"

"NORMAL"

"POINT"

"PLANE-LINE"

"LOCAL"

Commands used for calculation of weight and center-of-gravity location of the tile are:

"BLOCK"

"WEDGE"

"TETRA"

"FACE"

"CG"

A command to select a printer generated plot of geometry data is:

"VIEW"

The externally applied tile loads are specified by the commands:

"FORCE"

"MOMENT"

"PRESSURE"

"GLOAD"

The nonlinear stress analysis requires data given by the following commands:

"MATERIAL"

"IMPERFECT"

"PAD"

"CONTROL"

"NONSTRESS"

Generally, an input card image contains a command designator word followed by a list of data values for that command. The designator word must be enclosed by quotation marks ("). The list of data values can be continued on more than one card image. A slash (/) is needed to terminate a list of data values whose length is case dependent such as the number of nodes used to define a SIP boundary. The data must be input in a logical sequence or order. For example, before a node point can be referenced in a command, it must have been previously defined. Hence, the "NODE" data must precede the "CORNER" data.

DESCRIPTION OF INPUT COMMANDS

A detailed description of the input commands is given in this section in the general order that they would logically occur in an input deck.

Case Identification

Command: "TITLE"

Purpose: To input a card containing an identification title which is printed as a heading on various pages of an analysis output listing.

Format: "TITLE"
THIS IS AN EXAMPLE TITLE CARD

Description: The card following the "TITLE" command contains alphanumeric information in the first 72 columns.

Command: "CASEID"

Purpose: To input a card containing case identification information to be output on a file of calculated stress values. These values are required by the program for identification purposes on a special output file, TAPE2, but have no other effect on the program.

Format: "CASEID" NPART XMACH Q ALPHA

Description: This is a single card containing the following values:

NPART - Integer tile part number

XMACH - Mach number

Q - Dynamic pressure

ALPHA - Angle of attack

Command: "COMMENT"

Purpose: To allow users to insert comments directly into the input deck. These comments are printed in the output listing as they are encountered and have no other effect on the operation of the program.

Format: "COMMENT" N

$\left.\begin{array}{l}\text{———}\\ \text{———}\\ \text{———}\end{array}\right\}$ N comment cards

Description: The integer N specifies the number of comment cards, containing alphanumeric information, which are to be input following the "COMMENT" command.

Geometry Description

Command: "NODE"

Purpose: To input nodal coordinates used for the geometric definition of the TPS model.

Format: "NODE" N X Y Z

Description: This command contains the following values:

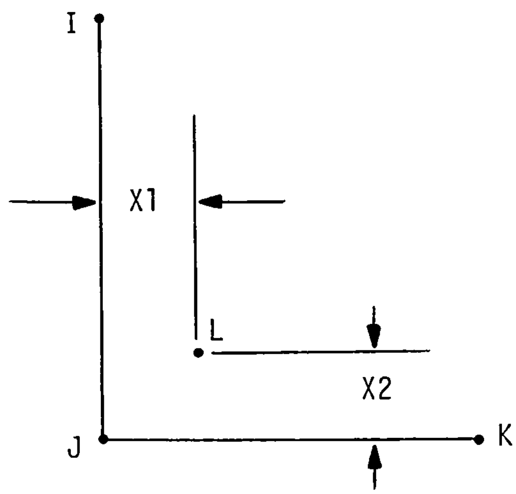
N - Integer node number assigned by user. Maximum value which can be used is 200.

X,Y,Z - The x,y,z coordinates of a node point, respectively.

Command: "CORNER"

Purpose: To establish a node point in the corner of a plane through previously defined node points. This command is often used to define SIP boundary nodes on the inner surface of the tile.

Format: "CORNER" I J K L X1 X2

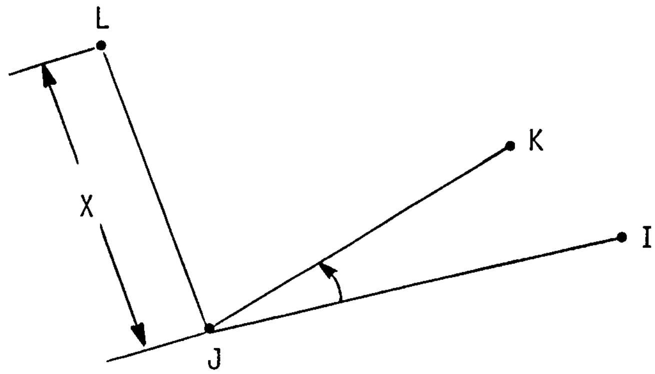


Description: This command is used to calculate the coordinates of a node, L, at given perpendicular distances X1 and X2 from the edges of a corner I-J-K.

Command: "NORMAL"

Purpose: To establish a node point at a given distance normal to a plane.

Format: "NORMAL" I J K L X

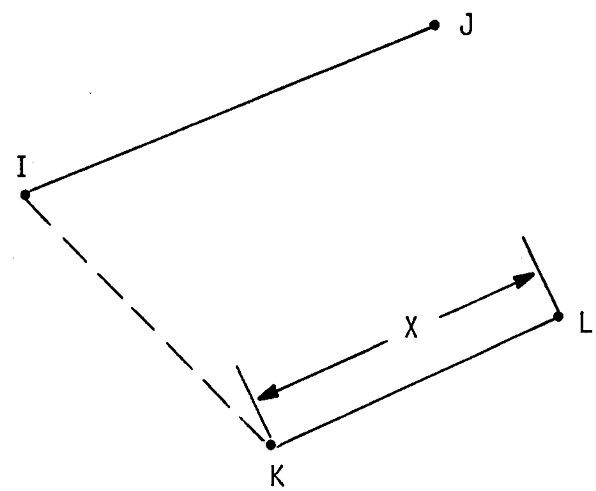


Description: The coordinates of a point, L, are calculated at a perpendicular distance, X, from plane I-J-K. The direction of $\overrightarrow{J\text{-}L}$ is given by the right-hand rule: $\overrightarrow{J\text{-}I} \times \overrightarrow{J\text{-}K}$.

Command: "POINT"

Purpose: To establish the position of a node point relative to previously defined nodes.

Format: "POINT" I J K L X

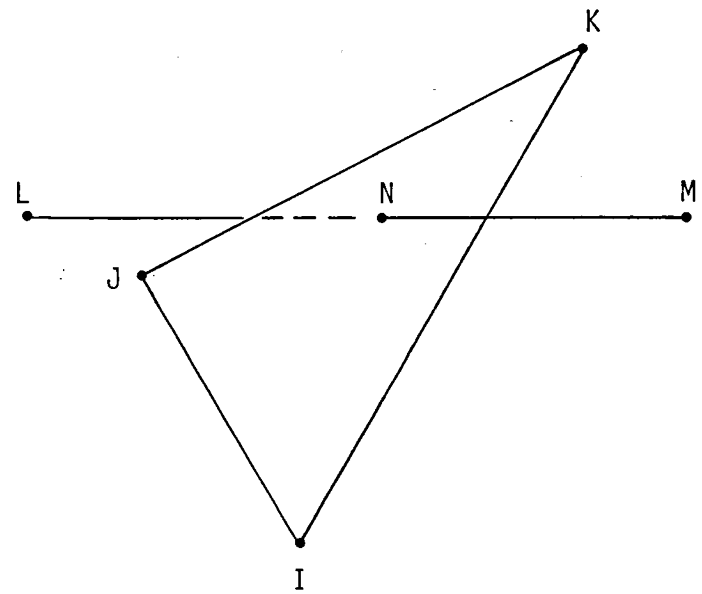


Description: The coordinates of a node, L, are calculated at a distance, X, from node, K, in the direction $\overrightarrow{I\text{-}J}$.

Command: "PLANE-LINE"

Purpose: To establish a node point at the intersection of a previously defined plane and line.

Format: "PLANE-LINE" I J K L M N



Description: The coordinates of a node, N, are calculated at the intersection of plane I-J-K and line L-M.

Command: "LOCAL"

Purpose: To print the coordinates of all node points with respect to a user specified local coordinate system.

Format: "LOCAL" L M N

Description: This command contains the following values:

L - A node defining the origin of the local coordinate system.

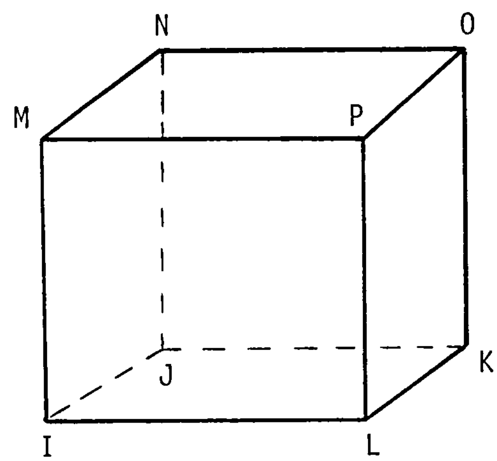
M - A node defining the direction of the local y-axis, $\overrightarrow{L\text{-}M}$.

N - A node located in the local x-y plane.

Command: "BLOCK"

Purpose: To input the node numbers at the corners, and the density, of a block element used for modeling a tile. The sides of the block need not be parallel but they must be planar.

Format: "BLOCK" I J K L M N O P / DENSITY

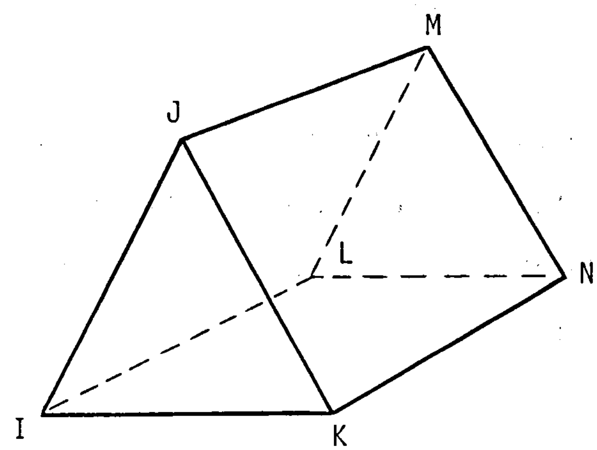


Description: The eight integer values I through P are previously defined node numbers at the corners of a block listed in the order as shown in the sketch. DENSITY is the weight per unit volume of the block.

Command: "WEDGE"

Purpose: To input the node numbers at the corners, and the density, of a wedge element used for modeling a tile. Multiple BLOCK, TETRA, and WEDGE commands can be used to build up irregular shaped tiles.

Format: "WEDGE" I J K L M N / DENSITY

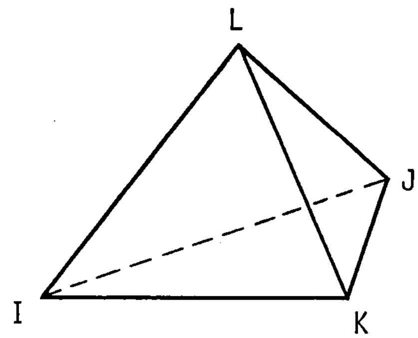


Description: The six integer values I through N are previously defined node numbers at the corners of the wedge. The triangular faces need not be parallel but the quadrilateral faces must be planar. DENSITY is given as weight per unit volume.

Command: "TETRA"

Purpose: To input the node numbers at the corners, and the density, of a tetrahedron element used for modeling a tile.

Format: "TETRA" I J K L / DENSITY

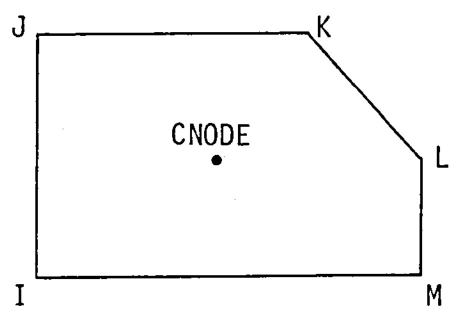


Description: The four integer values I through L are previously defined node numbers at the corners of the tetrahedron. DENSITY is given as weight per unit volume.

Command: "FACE"

Purpose: To input the node numbers at the corners, and the density, of a face element used for modeling the coating on a tile. Each face element must be planar.

Format: "FACE" I J K . . . / THICKNESS DENSITY CNODE



Description: Up to 50 previously defined node numbers may be used to specify the corners of a face element.

THICKNESS gives the thickness of the face.

DENSITY is given as weight per unit volume.

CNODE is a node number whose coordinates are established at the centroid of the face by the face command.

Command: "CG"

Purpose: To establish a node at the center of gravity of the tile and cause this c.g. location and tile weight to be printed.

Format: "CG" NODE

Description: This command is used after the tile has been defined as a composite of BLOCK, TETRA, WEDGE, and FACE commands. The node, NODE, is established at the center of gravity of the tile.

Command: "VIEW"

Purpose: To generate a plot of the tile model for use in checking input geometry. This plot is produced by the printer in the output listing.

Format: "VIEW" L M N

Description: The viewing plane for the plot is established by the previously defined nodes L, M, and N.

L - A node defining the origin of the viewing, x-y, plane.

M - A node defining the direction of the y-axis, L-M.

N - A node located in the x-y plane.

Tile Loads

Loading is applied to a tile by combinations of FORCE, MOMENT, PRESSURE, and GLOAD commands. A load case number is specified on each of these commands and all load contributions for commands with the same load case number are summed to form a total load case. Up to fifty load cases can be defined.

Command: "FORCE"

Purpose: To input a concentrated force at an existing node point.

Format: "FORCE" LDCASE NODE FX FY FZ

Description: A concentrated force with components FX, FY, and FZ along the X,Y,Z axes of the global coordinate system is applied at node point, NODE, and used in load case, LDCASE.

Command: "MOMENT"

Purpose: To input a concentrated moment at an existing node point.

Format: "MOMENT" LDCASE NODE MX MY MZ

Description: A concentrated moment with components MX, MY, and MZ about the X,Y,Z axes of the global coordinate system is applied at node point, NODE, and is used in load case, LDCASE.

Command: "PRESSURE"

Purpose: To input a pressure loading on a tile surface.

Format: "PRESSURE" LDCASE L PL M PM N PN

Description: A linear pressure distribution is defined over a triangular surface with corner node points L, M, and N. The pressure over the triangle varies linearly between pressures PL, PM, and PN which are specified at the corners with the direction of positive pressure given by $\overrightarrow{L\text{-}M} \times \overrightarrow{L\text{-}N}$. This pressure loading is used in load case, LDCASE.

Command "GLOAD"

Purpose: To input an acceleration or g-load factors at the tile center of gravity.

Format: "GLOAD" LDCASE GX GY GZ

Description: The g-loading is specified as components GX, GY, and GZ in the global coordinate system.

Nonlinear Analysis Data

The nonlinear analysis procedure requires the geometrical definition of the SIPS and filler bars which are defined by "PAD" commands and their nonlinear material curves defined by "MATERIAL" commands. Substrate deformation shapes are defined using "IMPERFECT" commands and the load case combination to be used for a particular analysis is given on the "NONSTRESS" command which causes the nonlinear analysis to be performed. A "CONTROL" command can precede the "NONSTRESS" command to reset parameters which govern execution of the analysis program. An index value must be assigned by the user to each material and each imperfection. These index values are referenced on the "PAD" commands to define the material and imperfections to be associated with each pad.

Command: "MATERIAL"

Purpose: To input the stiffness characteristics of the SIP and/or filler bar material. This input is in the form of curves which relate stresses to displacements or strains in the material. Curves for the SIP material must be specified for both normal and shear behavior. Filler bars cannot transmit any tensile or shear forces since they are not attached to the tile. Therefore, forces are only produced by the compression portion of the normal curve with the tensile portion represented by a horizontal line at zero stress and a shear curve is not required. All material curves are extrapolated as straight lines through the last two points on each extremity of the curve for cases with displacements beyond the bounds of the specified curve. Up to twenty different material curves can be defined. Three format options are available for input of material characteristics:

(1) Input of a table of points on a stress-displacement curve

(2) Select a material curve from a previously defined data file

(3) Input of linear material characteristics.

Format (1): "MATERIAL" INDEX "TABLE" T NPTS

Description: This form of the "MATERIAL" command is used for direct input of a table of points to define either a normal or shear stress-displacement curve.

INDEX - An integer 1 → 20 used to identify a particular material curve.

"TABLE" - Indicates a table of points follows.

T - Is the thickness of the material.

NPTS - The number of points on the curve which follow this "MATERIAL" command. One point is given per row in the table with the displacement value given first and the stress value given second.

The definition of the curve should start with the point having the largest negative displacement value and proceed to largest positive displacement.

Example:

```
"MATERIAL"    5    "TABLE"    0.160    6

    -.04            -15.0
    -.02475         - 6.0
    -.005           - 1.0
    0.0               0.0
     .02178           5.0
     .04410          15.0
```

Format (2): "MATERIAL" INDEX "NORMAL" or "SHEAR" "A1" "A2" N3 N4

Description: This form of the "MATERIAL" command is used to select a stress-displacement curve from a previously defined data file. The curves are given a unique four word identifier. The procedure used to establish or add curves to the required data file is given in Appendix A.

INDEX	- An integer 1 → 20 used to identify a particular material. This index is referred to on the "PAD" command to designate its material.
"NORMAL" or "SHEAR"	- "NORMAL" indicates the material is to be used for normal stress calculation. "SHEAR" indicates the material is to be used for shear stress calculation.
"A1" and "A2"	- Are the first and second words in the identifier name containing up to four characters and beginning with an alphanumeric character.
N3 and N4	- Are the third and fourth words in the identifier name containing up to four integer characters.

Example curve names:

"FBN"	"N16C"	1000	0
"SIPN"	"N16T"	1580	100
"SIPN"	"R09V"	0	0

Format (3): "MATERIAL" INDEX "LINEAR" SLOPE T

Description: This form of the "MATERIAL" command is used to define a linear stress-strain relation for use in analysis. The line defined by this command passes through the origin of a stress-strain plot and has the slope which is specified.

INDEX - An integer 1 → 20 used to identify a particular material curve.

"LINEAR" - Indicates the material has a linear stress-strain relationship.

SLOPE - Is the slope of the stress-strain curve (e.g., E or G).

T - Is the thickness of the SIP material (e.g., .09 or .16).

Command: "IMPERFECT"

Purpose: To define imperfections such as mismatch, warpage, or substrate deflection to be used in analysis. If no "IMPERFECT" commands are input, the inner tile surface and the substrate surface are taken to be flat.

Format: "IMPERFECT" INDEX ITYPE I J AIMP GAMY GAMX

Description: Imperfections are applied to only those SIPS and/or filler bars in which they are referenced on the "PAD" commands. Up to twenty different imperfections may be defined with the imperfection used in a particular analysis being the sum of the imperfections specified on the "NONSTRESS" command. The command contains the following values:

INDEX - An integer ($1 \rightarrow 20$) used to give an identification number to each imperfection.

ITYPE - An integer ($1 \rightarrow 6$) used to specify an imperfection type. The available types are:

= 1 For cylindrical imperfection

= 2 For spherical imperfection

= 3 For single cosine

= 4 For double cosine

= 5 For step function

= 6 For ramp function

I - Locates the origin of the local coordinate system of the imperfection.

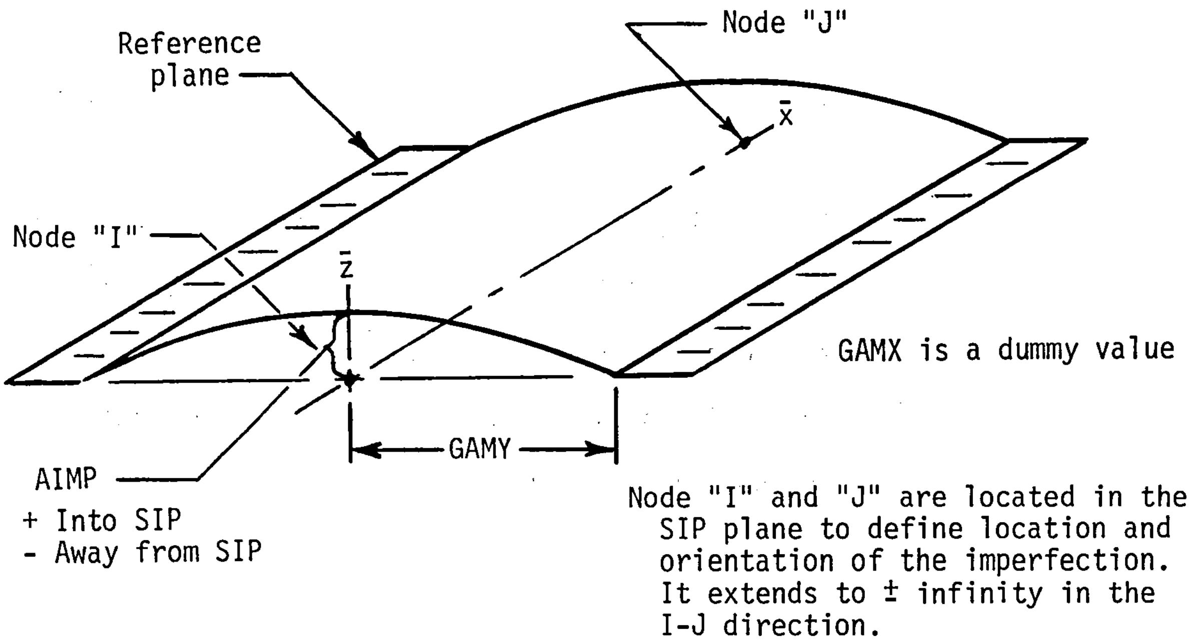
J - Defines the direction of the local x-axis, $\overrightarrow{I\text{-}J}$.

AIMP - Gives the imperfection amplitude; a positive value indicates a direction into the SIP and a negative value is away from the SIP.

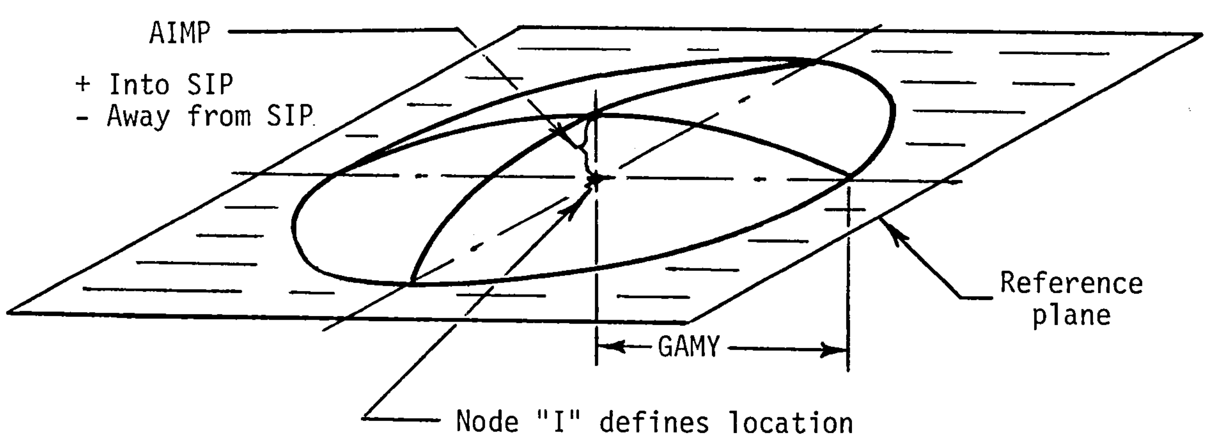
GAMY, GAMX - Wavelength parameters associated with the imperfections.

The meaning of these values that are applicable for defining each type of imperfection is given on the following figures. Some values are not required in the definition of some types of imperfections, but they must be input as dummy values.

ITYPE = 1 for cylindrical imperfection

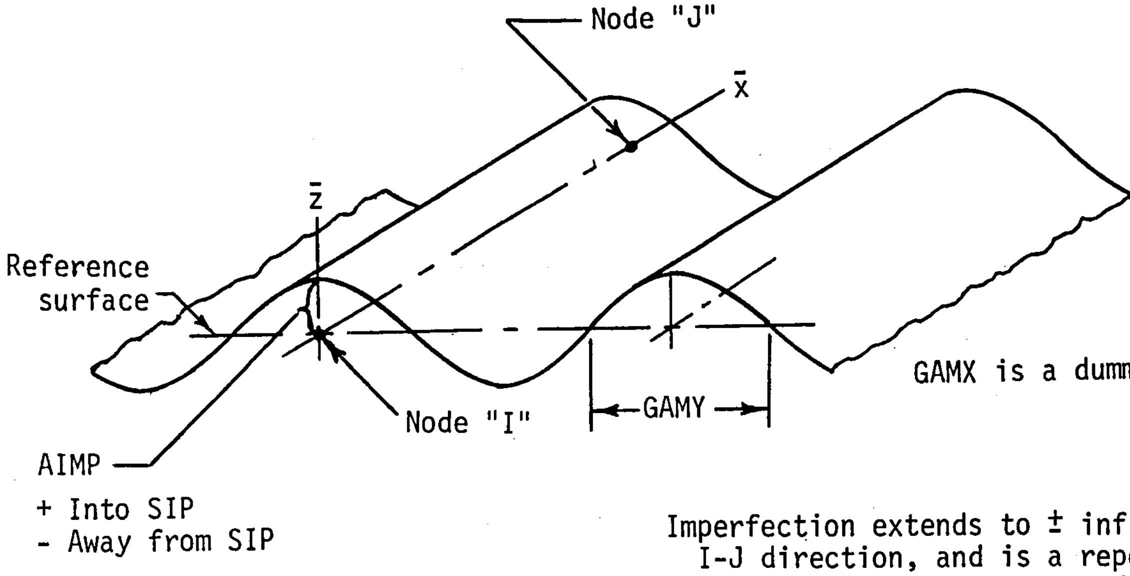


ITYPE = 2 for spherical imperfection



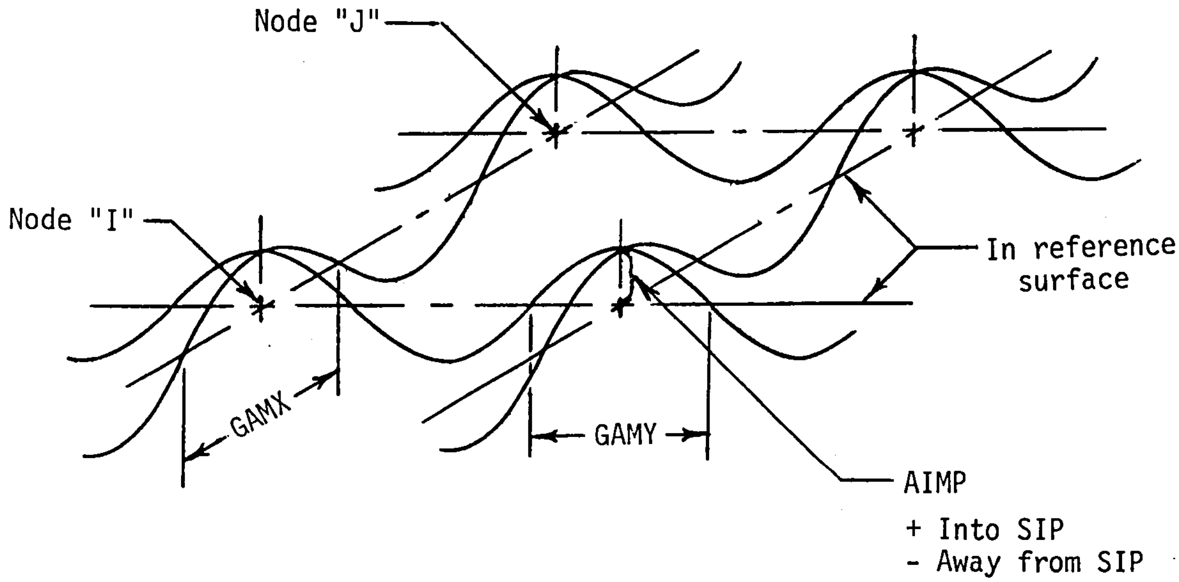
Node "J" and GAMX are dummy values.

ITYPE = 3 for single cosine

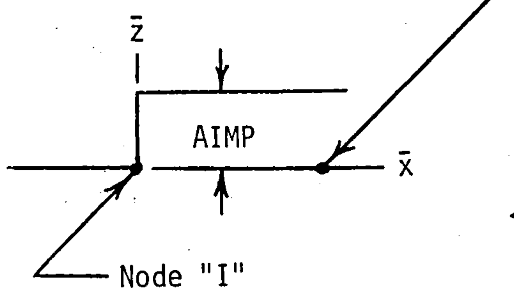


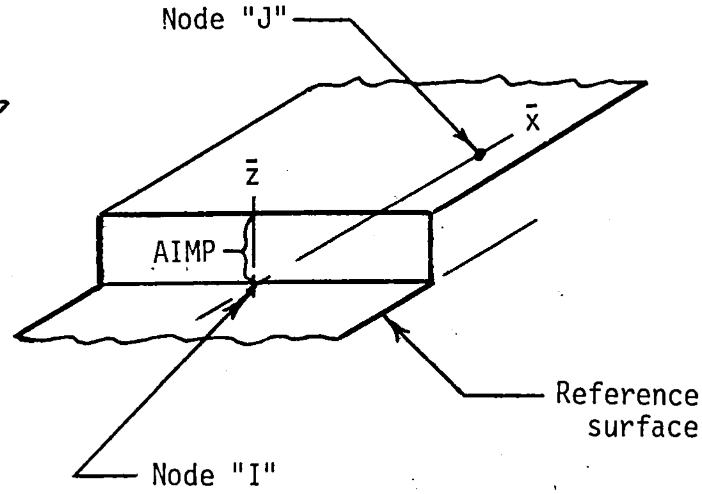
Imperfection extends to ± infinity in I-J direction, and is a repeating cosine in direction perpendicular to I-J.

ITYPE = 4 for double cosine



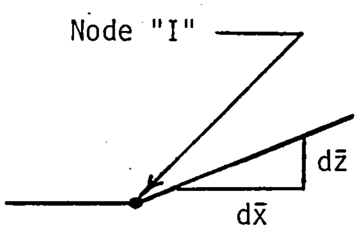
ITYPE = 5   for step function

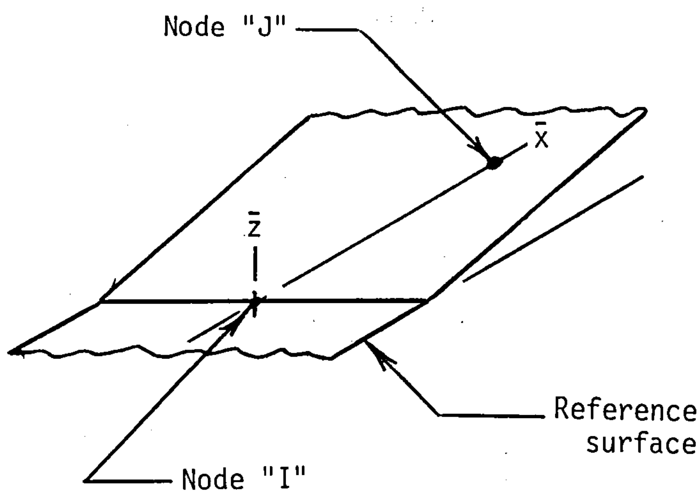




GAMY and GAMX are dummy values.

ITYPE = 6   for ramp function





$$AIMP = \frac{d\bar{z}}{d\bar{x}}$$

GAMY and GAMX are dummy values.

Command: "PAD"

Purpose: To input information needed to describe a pad of SIP or filler bar. Multiple pads can be defined to model tiles having a nonplanar attachment surface or to provide for a SIP with a variation in material properties. The variation could be modeled as a series of discrete pads.

Format: "PAD" INDEX "SIP" MATN MATS I J K .../IMP1 IMP2 .../
or
"FB"

Description: Up to twenty pads can be defined and each command contains the following values:

INDEX - An integer 1 → 20 used to identify a particular pad for reference in the output listing.

"SIP" or "FB" - "SIP" indicates the pad is SIP material. "FB" indicates the pad is filler bar material.

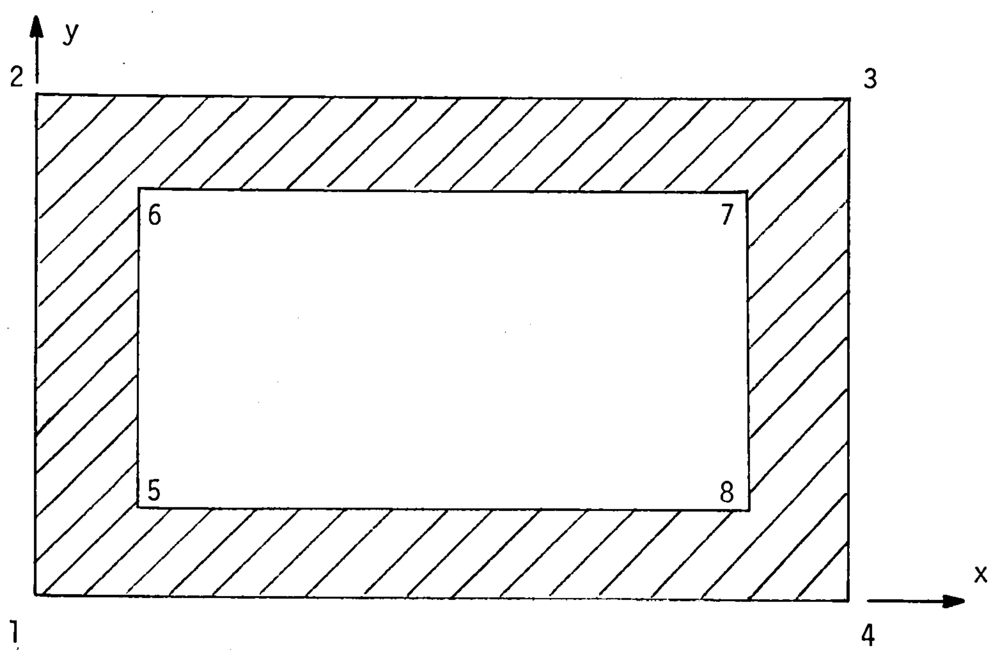
MATN - An integer 1 → 20 referring to an entry defined by a "MATERIAL" command. The corresponding material properties are used for normal stress calculations on this pad.

MATS - Similar to MATN, but refers to material properties for shear stress calculations. For filler bars, this entry should be zero.

I J K.../ - A list of nodes which define the boundary of the pad. The first three nodes in the list set up a local coordinate system for the pad. The origin is located at the first node and the Y-axis passes through the second node. The third node determines the direction of the X-axis (i.e., which side of the Y-axis it will be).

The nodes around the outside of a pad should be input in a clockwise direction so that the local Z-axis points into the tile. For a pad with a hole, such as a filler bar, the inside boundary should be input in a counterclockwise direction to subtract out the contribution of the interior hole. Up to two hundred nodes can be input in this list which must be terminated with a slash.

An example of numbering a filler bar follows:



Numbering:

1 2 3 4 1 5 8 7 6 5 /

IMP1 IMP2.../ - A list of integers 1 $\rightarrow$ 20 referring to entries defined by "IMPERFECT" commands. These imperfections will be the ones applied to this particular pad if referenced on a "NONSTRESS" command. If no imperfections are to be considered for the pad, a zero must be input with the list terminated with a slash.

Command: "CONTROL"

Purpose: The "CONTROL" command is used to define parameters which control the nonlinear stress analysis procedure and the resulting printed output.

Format:

"CONTROL"	IPRT	NDIV	NSTEPS	MESH	TEST	NCON
Default values	0	1	5	10	.01	0

If a "CONTROL" command is not input, the default values shown above will be used.

Description: An iterative solution procedure is used to determine the tile displacements which give equilibrium between the applied loads and reaction of the integrated material stresses in the pads. The accuracy of solution and required computational time are controlled by the values of NSTEPS, MESH, and TEST. The quantity of printed output is controlled by the values of IPRT, NDIV, and NCON.

During the solution procedure, the total applied tile load, which is specified as the combination of load cases on a "NONSTRESS" command, can be applied as a sequence of equal load increments or steps. A converged solution is obtained at each load step. The following values control this solution process:

IPRT - = 0 Print iteration history and resulting stresses at the final load step which is the total applied load.

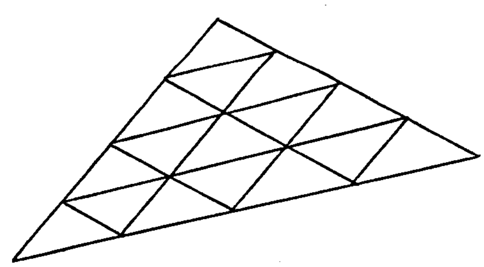
= 1 Print iteration history at each of NSTEPS load steps and print stresses for only the final load step.

= 2 Print both iteration history and resulting stresses at each load step.

NDIV - Specifies the number of points along edges of pads at which stresses will be calculated and printed.

NSTEPS - Number of load steps to be used in the solution process. A larger number of steps requires less iterations per step to reach convergence. The number of load steps which will result in minimum computational time is problem dependent. For efficiency, NSTEPS should be unity when only linear materials are used.

MESH - Defines the refinement of a mesh of subtriangles used for numerical integration of stresses over a triangular region of a SIP or filler bar. Each side of the triangular region is divided into "MESH" equal intervals in forming the subtriangles. A linear distribution of stress is assumed over each subtriangle in the mesh. Therefore, greater accuracy but longer computation time results from a finer mesh. An example of the subtriangle pattern for MESH = 4 is shown below.



TEST - The convergence test parameter for the iterative solution procedure. The iterations are terminated when the difference between each component (F_x, F_y, F_z, M_x, M_y, M_z) of the externally applied loads and pad material reactions is less than the value of TEST.

NCON - Number of intervals along both axes of a rectangular grid over each pad at which stresses will be printed. Up to 34 intervals may be specified. If NCON is specified as zero, these stress distributions will not be printed. The boundary of a rectangular grid is calculated from the minimum and maximum local x and y coordinate values of the corner points of the pad. If the pad itself is not rectangular, values on the grid outside the pad boundary will be printed, but should not be mistakenly interpreted as SIP stresses. Only values inside the pad boundary are SIP stresses.

Command: "NONSTRESS"

Purpose: The "NONSTRESS" command causes execution of the nonlinear analysis procedure.

Format: "NONSTRESS" L1 L2 ... / IMP1 IMP2 ... /

Description: The analysis will include the load case combination L1 + L2 + ... and imperfections IMP1+IMP2+ ... This command may be executed multiple times during a single run for different load conditions and imperfection combinations.

EXAMPLE ANALYSIS

This section contains input data and corresponding calculated results for the analysis of an example tile and is designed to illustrate use of all input commands. This example tile is representative of an acreage tile having a 6-inch square planform and a thickness of 2 inches. A graphical representation of the tile geometry is shown in figure 2. The tile could have been modeled using a single block element but is divided into a block, a wedge, and three tetrahedrons to illustrate use of various geometry input commands. The tile is mounted on a single SIP surface with corner nodes numbered 5, 6, 7, and 8. The SIP is surrounded by a filler bar with a width of 1/2 inch.

The loads applied to the tile include a concentrated force and moment, a pressure gradient on the outer surface, and a statically equivalent acceleration or g-load. These loads are set up to be symmetric about the diagonal, Nodes 1 and 3, of the tile. A nonlinear material curve is input for the SIP in the normal or through-the-thickness direction and linear properties are specified in shear. Filler bar properties are defined on a separate data file as illustrated in appendix A. A nonlinear analysis is performed for a combination of all four load cases and the sum of a cylindrical and double cosine imperfection. The input commands for this example tile analysis follows:

```
"TITLE"
            ACREAGE TILE EXAMPLE
"CASEID"    111222333  1.05  635.3  3.83
"COMMENT"   3
            THIS EXAMPLE IS DESIGNED TO ILLUSTRATE
            USE OF ALL AVAILABLE INPUT COMMANDS.
            PREPARED BY GARY L. GILES      3-23-81.
"NODE"       1    0.0    0.0    0.0
"NODE"       2    0.0    6.0    0.0
"NODE"       3    6.0    6.0    0.0
"NODE"       4    6.0    0.0    0.0
"CORNER"     4     1      2      5    0.5    0.5
"CORNER"     1     2      3      6    0.5    0.5
"CORNER"     2     3      4      7    0.5    0.5
"CORNER"     3     4      1      8    0.5    0.5
"NORMAL"     4     1      2      9    2.0
"NORMAL"     1     2      3     10    2.0
"NORMAL"     2     3      4     11    2.0
"NORMAL"     3     4      1     12    2.0
"POINT"      1     4      1     13    3.0
"POINT"      9    12      9     14    3.0
"POINT"     10    11     10     15    3.0
"PLANE-LINE" 13   14     15      2     3     16
"LOCAL"      1     2      3
"BLOCK"      1     2     16     13     9     10    15    14 / .005208
"WEDGE"      3     4     11     16    13     15 / .005208
"TETRA"     11    13     14     15 / .005208
"TETRA"      4    11     14     12 / .005208
"TETRA"      4    11     14     13 / .005208
"FACE"       9    10     11     12 / .011    .061    17
"FACE"       1     2     10      9 / .011    .061    18
"FACE"       2     3     11     10 / .011    .061    19
"FACE"       3     4     12     11 / .011    .061    20
"FACE"       4     1      9     12 / .011    .061    21
"CG"        22
"VIEW"       1     2      3
"FORCE"      1    17      5.0     5.0     0.0
"MOMENT"     2     1    -15.0    15.0     0.0
"PRESSURE"   3     9      1.5     12      1.0     10     1.0
"PRESSURE"   3    10      1.0     12      1.0     11     0.5
"GLOAD"      4       0.0      0.0    50.0
```

```
"MATERIAL"  1  "TABLE"   .160   43
     -.09000  -12.00000
     -.08800  -10.00000
     -.08200   -8.00000
     -.07500   -6.00000
     -.06625   -4.68000
     -.06355   -4.26000
     -.06068   -3.84000
     -.05713   -3.39000
     -.05305   -2.95000
     -.04860   -2.53000
     -.04288   -2.08000
     -.03590   -1.62000
     -.02963   -1.29000
     -.02495   -1.07000
     -.01970    -.85000
     -.01640    -.73000
     -.01310    -.61000
     -.01060    -.53000
     -.00738    -.42000
     -.00468    -.32000
     -.00303    -.26000
     -.00148    -.18000
     -.00058    -.11000
     0.00000    0.00000
      .00600     .25000
      .01100     .50000
      .01500     .83000
      .01800    1.12000
      .02200    1.64000
      .02700    2.44000
      .03200    3.63000
      .03600    5.10000
      .04000    6.80000
      .04600    9.10000
      .05000   10.40000
      .05500   11.70000
      .06000   12.80000
      .06300   13.40000
      .06500   13.40000
      .06600   13.30000
      .06800   12.75000
      .07000   11.30000
      .08000    0.00000
"MATERIAL"     2  "NORMAL"  "FBN"   "N16C"  1000  0
"MATERIAL"     3    "LINEAR"    9.0    .16
"IMPERFECT"    1    3    5    7    0.02    12.0    0.0
"IMPERFECT"    2    4    5    7   -0.0075  12.0    2.0
"PAD"          1  "FB"  2  0  1  2  3  4  1  5  8  7  6  5 / 1  2 /
"PAD"          2  "SIP"  1  3  5  6  7  8 / 1  2 /
"CONTROL"      2   2   2   15  .001   10
"NONSTRESS"    1   2   3   4 / 1   2 /
```

The printed output listing is generated on the file, TAPE6. Selected portions of this output listing for the example tile will be presented and described. Portions of the output which are self-explanatory have been omitted for brevity. Also, portions which contain the same type of information as the included portions but are for different load steps and material pads have been omitted.

As input commands defining geometry, loads, and materials are read, they are printed in the output along with some self-explanatory, calculated information such as element weight and center-of-gravity location. This portion of the listing corresponds to all commands in the example input listing except for the "NONSTRESS" command and is not included herein.

The "NONSTRESS" command initiates execution of the nonlinear stress analysis of the model defined by all preceding commands. Values for each of the defined load cases, referenced to the tile center of gravity, are printed as shown on the following page. These load vectors correspond to F_X, F_y, F_Z, M_x, M_y, M_Z in the global coordinate system. Next, the loads and imperfections which are to be combined for a particular analysis are printed with the corresponding total applied tile loads.

NONSTRESS
RESOLUTION OF EXTERNAL LOADS TO TILE C.G.

1	5.000	5.000	0.000	-4.720	4.720	-.000
2	0.000	0.000	0.000	-15.000	15.000	0.000
3	0.000	0.000	36.000	-9.000	9.000	0.000
4	0.000	0.000	21.567	0.000	0.000	0.000

ACREAGE TILE EXAMPLE

NONLINEAR STRESS ANALYSIS - NASA,LRC

LOADS COMBINED 1 2 3 4

SELECTED IMPERFECTIONS= 1, 2,

EXTERNAL LOADS = .50000E+01 .50000E+01 .57567E+02 -.28720E+02 .28720E+02 -.71054E-13

During the solution procedure, the total external tile load can be applied as a sequence of equal load increments or steps. These load steps are defined in the program by multiplying the total load, shown on the previous page, by a factor, lambda. An analysis is performed for the unloaded tile, lambda = 0.0, and in equal steps up to the total load, lambda = 1.0. A converged solution is obtained at each load step. At each load step or value of lambda, the solution iteration history can be printed if selected by the value IPRT on the "CONTROL" command. The iteration history contains the values of displacements and rotations of the tile center of gravity (QCG) and corresponding values of the difference between the applied forces and pad material reactions (DELF). The iteration histories for the acreage tile example at the initial and final load steps are shown on the following page. The initial set of iterations, which are numbered one, for lambda equal zero correspond to reaching equilibrium with no applied loads or imperfections. This resulting equilibrium position is solely a function of the material stress displacement curves and allows curves which do not pass through the origin to be used. This position is taken to be the reference location from which all displacement values are calculated herein. The remaining iterations at lambda equal zero seek equilibrium for the specified substrate deflections. All remaining iteration histories are for applied loads.

ACREAGE TILE EXAMPLE

```
   LAMBDA=      0.000

ITER=    1  QCG= 0.             0.             0.             0.             0.             0.
            DELF= 0.            0.             .15987E-13   -.40390E-26    .55536E-26   0.

ITER=    1  QCG=  .12425E-29    .11350E-29     .30161E-17   -.10748E-29    .11766E-29   -.31862E-57
            DELF= 0.            0.             .15987E-13   -.40390E-26    .55536E-26   0.

ITER=    1  QCG=  .12425E-29    .11350E-29     .30161E-17   -.10748E-29    .11766E-29   -.31862E-57
            DELF= -.31529E-40  -.31529E-40     .29339E+02    .86509E+00   -.86509E+00    .48940E-54

ITER=    2  QCG= -.85459E-04   -.85459E-04     .19333E-01    .80927E-04   -.80927E-04   -.24652E-31
            DELF=  .18296E-14   .30493E-14     .11918E+01   -.32297E+00    .32297E+00   -.27768E-27

ITER=    3  QCG=  .13282E-05    .13282E-05     .20186E-01   -.12578E-05    .12578E-05   -.17256E-30
            DELF= -.77186E-15  -.71468E-15     .71663E-01   -.11282E+00    .11282E+00    .10855E-26

ITER=    4  QCG=  .40868E-04    .40868E-04     .20248E-01   -.38700E-04    .38700E-04    .12326E-31
            DELF= -.30493E-15   .30493E-15    -.32761E-02    .30020E-02   -.30020E-02   -.10729E-27

ITER=    5  QCG=  .39820E-04    .39820E-04     .20246E-01   -.37708E-04    .37708E-04   -.44296E-31
            DELF= -.30493E-15   .78921E-41    -.27759E-05   -.35099E-05    .35099E-05    .56798E-28
  .
  .
  .
   LAMBDA=      1.000

ITER=    1  QCG=  .54960E-02    .54960E-02     .34804E-01   -.35210E-02    .35210E-02    .17661E-16
            DELF=  .25000E+01   .25000E+01     .28784E+02   -.14360E+02    .14360E+02    .56843E-13

ITER=    2  QCG=  .86570E-02    .86570E-02     .43676E-01   -.48309E-02    .48309E-02    .45023E-16
            DELF= -.25580E-12  -.22737E-12    -.99901E+01    .62582E+01   -.62582E+01    .24158E-12

ITER=    3  QCG=  .83247E-02    .83247E-02     .41957E-01   -.45162E-02    .45162E-02    .86253E-16
            DELF= -.17053E-12  -.17053E-12    -.41643E+00    .20044E+00   -.20044E+00   -.75318E-12

ITER=    4  QCG=  .83165E-02    .83165E-02     .41873E-01   -.45084E-02    .45084E-02   -.42289E-16
            DELF=  .56843E-13  -.28422E-13    -.87313E-03    .20720E-03   -.20720E-03    .78160E-12
```

Stress information is always printed for the total applied load, lambda equal 1.0, and can be optionally selected at each of the load steps. This stress output is printed separately for each pad and can be identified by the user defined pad number. The output for the SIP, pad 2, for the example tile is given on the following page. The material curve indicators used for normal and shear calculations are printed with imperfection indicators and the applied load cases. Forces in the pad local coordinate system which are equivalent to the pad stresses are then output.

Next, a table of stress related data which are computed during the analysis is printed. The information is printed at the corners of the pad, identified by node number, and any intermediate locations on the pad edge which were selected by NDIV on the "CONTROL" command.

The coordinates of the locations where information is to be output are given by the values XX and YY in the pad local coordinate system. Displacements at the inner surface of the tile (XDISP, YDISP, and ZDISP) are given in the pad local coordinate system relative to the equilibrium condition with no imperfections and no externally applied tile loads. Shear stresses, TAUXZ and TAUYZ, are given in the local X-Z and Y-Z planes, respectively. The normal or through-the-thickness stress in the pad is denoted as SIGZ. The total substrate deflection, WIMP, is the sum of the "IMPERFECT" contributions and the total normal displacement of the pad material relative to the origin of the stress-displacement curve is given by WEXT. The values DTAUXZ, DTAUYZ, and DSIGZ are the slopes or derivatives of the corresponding stress-displacement curves for the displacement value at that particular pad location.

ACREAGE TILE EXAMPLE

PAD NUMBER 2 *SIF *
MATERIAL - NORMAL = 1
MATERIAL - SHEAR = 3
IMPERFECTIONS = 1, 2,
LOADS COMBINED= 1, 2, 3, 4,
RESOLUTION OF FORCES ON PAD= .50000E+01 .50000E+01 .57638E+02 .11025E+03 -.11025E+03 .17653E-12

LOCAL COORDINATES

NODE	XX	YY	XDISP	YDISP	ZDISP	TAUXZ	TAUYZ	SIGZ
			WIMP	WEXT		DTAUXZ	DTAUYZ	DSIGZ
5	0.000	0.000	.35556E-02	.35556E-02	.64415E-01	.20000E+00	.20000E+00	.10898E+02
			.12500E-01	.51915E-01		.56250E+02	.56250E+02	.26000E+03
	0.000	2.500	.35556E-02	.35556E-02	.53144E-01	.20000E+00	.20000E+00	.29109E+01
			.24166E-01	.28978E-01		.56250E+02	.56250E+02	.23800E+03
6	0.000	5.000	.35556E-02	.35556E-02	.41873E-01	.20000E+00	.20000E+00	.40739E+01
			.86650E-02	.33208E-01		.56250E+02	.56250E+02	.36750E+03
	2.500	5.000	.35556E-02	.35556E-02	.30602E-01	.20000E+00	.20000E+00	.43139E+00
			.20974E-01	.96279E-02		.56250E+02	.56250E+02	.50000E+02
7	5.000	5.000	.35556E-02	.35556E-02	.19331E-01	.20000E+00	.20000E+00	.69188E-02
			.19164E-01	.16605E-03		.56250E+02	.56250E+02	.41667E+02
	5.000	2.500	.35556E-02	.35556E-02	.30602E-01	.20000E+00	.20000E+00	.43139E+00
			.20974E-01	.96279E-02		.56250E+02	.56250E+02	.50000E+02
8	5.000	0.000	.35556E-02	.35556E-02	.41873E-01	.20000E+00	.20000E+00	.40739E+01
			.86650E-02	.33208E-01		.56250E+02	.56250E+02	.36750E+03
	2.500	0.000	.35556E-02	.35556E-02	.53144E-01	.20000E+00	.20000E+00	.29109E+01
			.24166E-01	.28978E-01		.56250E+02	.56250E+02	.23800E+03
5	0.000	0.000	.35556E-02	.35556E-02	.64415E-01	.20000E+00	.20000E+00	.10898E+02
			.12500E-01	.51915E-01		.56250E+02	.56250E+02	.26000E+03
	0.000	0.000	.35556E-02	.35556E-02	.64415E-01	.20000E+00	.20000E+00	.10898E+02
			.12500E-01	.51915E-01		.56250E+02	.56250E+02	.26000E+03

Displacement information is printed next for the total loading condition as shown on the following page for the acreage tile example. Displacement vectors, three translations, and three rotations are given at the tile center of gravity for the reference equilibrium position (lambda = 0 and no imperfections) and for the total applied load condition (lambda = 1.0 and including imperfections). A table of displacements, DELX, DELY, and DELZ, are then printed for each node point. Also included in the table are the user defined coordinates of the nodes. The displacement values correspond to the difference between the reference and final equilibrium positions. All values are given in the global coordinate system.

ACREAGE TILE EXAMPLE

FINAL NODAL DISPLACEMENTS IN GLOBAL SYSTEM

QCGREF AT LAMBDA=0,AND NO IMPERFECTIONS=

.12425E-29 .11350E-29 .30161E-17 -.10748E-29 .11766E-29 -.31862E-57

QCG AT LAMBDA=1,INCLUDING ANY IMPERFECTIONS=

.83165E-02 .83165E-02 .41873E-01 -.45085E-02 .45085E-02 .91104E-16

NODE	X	Y	Z	DELX	DELY	DELZ
1	0.0000	0.0000	0.0000	.0036	.0036	.0689
2	0.0000	6.0000	0.0000	.0036	.0036	.0419
3	6.0000	6.0000	0.0000	.0036	.0036	.0148
4	6.0000	0.0000	0.0000	.0036	.0036	.0419
5	.5000	.5000	0.0000	.0036	.0036	.0644
6	.5000	5.5000	0.0000	.0036	.0036	.0419
7	5.5000	5.5000	0.0000	.0036	.0036	.0193
8	5.5000	.5000	0.0000	.0036	.0036	.0419
9	0.0000	0.0000	2.0000	.0126	.0126	.0689
10	0.0000	6.0000	2.0000	.0126	.0126	.0419
11	6.0000	6.0000	2.0000	.0126	.0126	.0148
12	6.0000	0.0000	2.0000	.0126	.0126	.0419
13	3.0000	0.0000	0.0000	.0036	.0036	.0554
14	3.0000	0.0000	2.0000	.0126	.0126	.0554
15	3.0000	6.0000	2.0000	.0126	.0126	.0283
16	3.0000	6.0000	0.0000	.0036	.0036	.0283
17	3.0000	3.0000	2.0000	.0126	.0126	.0419
18	0.0000	3.0000	1.0000	.0081	.0081	.0554
19	3.0000	6.0000	1.0000	.0081	.0081	.0283
20	6.0000	3.0000	1.0000	.0081	.0081	.0283
21	3.0000	0.0000	1.0000	.0081	.0081	.0554
22	3.0000	3.0000	1.0560	.0083	.0083	.0419

A nonzero value of NCON on the "CONTROL" command is used to request printing of imperfection and stress values at points on a grid over each SIP pad. The boundary of this rectangular grid is calculated from the minimum and maximum X and Y values of corner nodes in the pad local coordinate system. If the pad is not rectangular, values on the grid outside the boundary will be printed but should not be mistakenly interpreted as SIP stresses.

The output grids of imperfection and stress data for the acreage tile example are shown on the following page. The origin of the local pad coordinate system corresponds to the value in the lower, left corner of the table with the local y-axis positioned vertically and the local x-axis positioned horizontally. The maximum and minimum stress values are calculated and printed on separate lines. For convenience of use at an interactive terminal, the stress grid and associated maximum and minimum stresses for each pad are output on a file named TAPE1.

When multiple tiles and/or multiple load conditions are analyzed during a single run, the information of primary interest is the maximum stress values that are calculated. Such information is output on a file named TAPE2 which contains a table of values with each line having information from the CASEID command, a load condition identifier, and a maximum stress value. Each line of the table is written as a tile part number, the load identifier, Mach number, dynamic pressure, angle of attack, and maximum stress as shown below for the acreage tile example.

```
111222333  1  1.05  635.3    3.83 10.8979
```

ACREAGE TILE EXAMPLE

IMPERFECTION GRID   SIP 2

LOADS COMBINED=   1,   2,   3,   4,
SELECTED IMPERFECTIONS= 1, 2,

.009	.008	.010	.012	.016	.021	.025	.026	.026	.023	.019
.012	.011	.010	.011	.014	.018	.023	.026	.027	.026	.023
.016	.014	.012	.011	.012	.015	.019	.023	.026	.027	.026
.020	.019	.016	.013	.012	.013	.015	.019	.023	.026	.026
.023	.023	.021	.017	.014	.013	.013	.015	.019	.023	.025
.024	.026	.025	.022	.018	.014	.013	.013	.015	.018	.021
.023	.026	.027	.025	.022	.018	.014	.012	.012	.014	.016
.020	.024	.027	.027	.025	.022	.017	.013	.011	.011	.012
.016	.021	.025	.027	.027	.025	.021	.016	.012	.010	.010
.014	.017	.021	.024	.026	.026	.023	.019	.014	.011	.008
.013	.014	.016	.020	.023	.024	.023	.020	.016	.012	.009

STRESS GRID   SIP 2

LOADS COMBINED=   1,   2,   3,   4,
SELECTED IMPERFECTIONS= 1, 2,

4.07	3.42	2.62	1.75	.96	.43	.15	-.07	-.21	-.18	.01
3.67	3.45	3.05	2.33	1.53	.81	.35	.10	-.17	-.23	-.18
3.19	3.12	3.07	2.76	2.15	1.40	.73	.31	.08	-.17	-.21
2.75	2.60	2.71	2.78	2.56	2.05	1.33	.70	.31	.10	-.07
2.59	2.25	2.26	2.43	2.61	2.45	2.01	1.33	.73	.35	.15
2.91	2.18	1.98	2.08	2.33	2.55	2.45	2.05	1.40	.81	.43
3.82	2.48	1.96	1.84	2.02	2.33	2.61	2.56	2.15	1.53	.96
5.84	3.46	2.32	1.88	1.84	2.08	2.43	2.78	2.76	2.33	1.75
8.15	5.54	3.36	2.32	1.96	1.98	2.26	2.71	3.07	3.05	2.62
9.94	8.04	5.54	3.46	2.48	2.18	2.25	2.60	3.12	3.45	3.42
10.90	9.94	8.15	5.84	3.82	2.91	2.59	2.75	3.19	3.67	4.07

MAXIMUM STRESS = 10.90
MINIMUM STRESS = -.23

END OF RUN

The tile part number, Mach number, dynamic pressure, and angle of attack are values input on the CASEID command. The load identifier is the first load case given on the NONSTRESS command and the last value is the corresponding calculated maximum stress.

The printed output described in this section is repeated for each NONSTRESS command which is input. Termination of the run is indicated by the END OF RUN message as shown on the bottom of the preceding page.

CONCLUDING REMARKS

User documentation is presented for a computer program which considers the nonlinear properties of the strain isolator pad (SIP) in the static stress analysis of the shuttle thermal protection system (TPS). This program is generalized to handle an arbitrary SIP footprint including cutouts for instrumentation and filler bar. Multiple SIP surfaces can be defined to model tiles in unique locations such as leading edges, intersections, and penetrations. The nonlinearity of the SIP is characterized by experimental stress-displacement data for both normal and shear behavior. Stresses in the SIP are calculated using a Newton iteration procedure to determine the six rigid body displacements of the tile which develop reaction forces in the SIP to equilibrate the externally applied loads.

This user documentation gives an overview of the analysis capabilities, a detailed description of required input data, and an example to illustrate use of the program.

APPENDIX A

MATERIAL CURVE DATA FILE

The pad material curves used in the analysis are a function of material thickness, orientation (normal or shear directions), and the history of loading the material has experienced. Thus, a large number of material curves may be required for use in an analysis effort. The SPAR data handling utilities, reference 4, can be used by the analysis program for convenient bookkeeping, storage, and retrieval of these material curves. The curves are stored on a file called SPARLA and each curve is referred to by a unique, four word name assigned by the user. An example of input data required to establish or add curves to the SPARLA data file is given in this appendix.

The input statements are free-field format and a detailed description of the statements used is given in reference 7. The example input which follows is for the filler bar curve which was used for the acreage tile example presented in the main body of this report.

```
[XQT  AUS
TABLE(NI=2,NJ=29)
FBN  N16C  1000  0
I=1,2
J=1,29
0.16              0.0
-.07876          -9.95000
-.07640          -8.96000
-.07490          -8.36000
-.07299          -7.69000
-.07106          -7.10000
-.06870          -6.46000
-.06677          -5.95000
-.06386          -5.32000
-.06120          -4.87000
-.05847          -4.35000
-.05537          -3.88000
-.05242          -3.49000
-.04909          -3.09000
-.04535          -2.72000
-.04106          -2.31000
-.03732          -2.01000
-.03284          -1.70000
-.02850          -1.41000
-.02423          -1.16000
-.02053           -.95000
-.01722           -.79000
-.01343           -.61000
-.01112           -.50000
-.00791           -.37000
-.00453           -.21000
-.00177           -.09000
0.00000           0.00000
 .01600           0.00000
TABLE(        )
    .
    .
    .
[XQT  DCU
TOC 1
PRINT 1
[XQT  EXIT
```

Each curve is given a four word name; the first two are alphanumeric and the last two are integers. The name of the curve in this example is FBN N16C 1000 0 as shown in the third statement. The values NI and I refer to the number of columns in the table which is two. Values NJ and J refer to the number of rows in the table which is twenty-nine for the example shown. The first row of the table contains the thickness of the pad and a dummy value (0.16 and 0.0 in the example). The remaining twenty-eight rows are points on the stress-displacement curve with the displacement value given first followed by the stress value. Such tables of data can be repeated for input of multiple curves as indicated. The last four statements shown in the example request the information on the data file to be printed for verification and termination of the run.

REFERENCES

1. Sawyer, James Wayne; and Rummler, Donald R.: Room Temperature Mechanical Properties of Shuttle Thermal Protection System Materials. NASA TM 81786, April 1980.

2. Housner, Jerrold M.; and Garcia, Ramon: Nonlinear Static TPS Analysis. NASA TM 81785, March 1980.

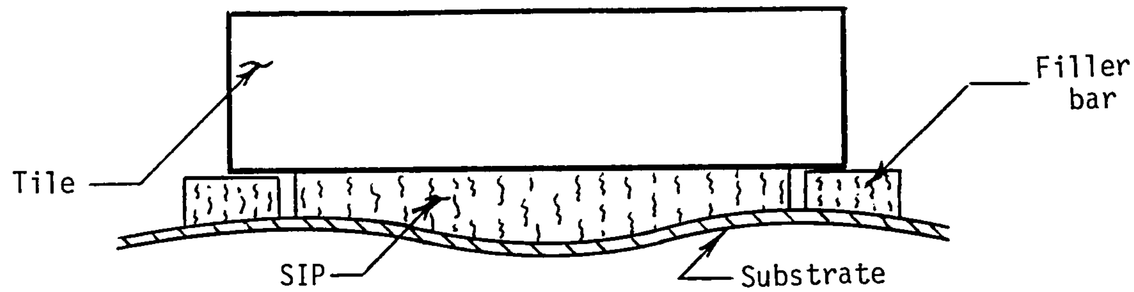
3. Zienkiewicz, O. C.: The Finite Element Method in Engineering Science. McGraw-Hill Book Co., Inc., c.1971, pp. 116-117.

4. Dobrowski, A. D.; and Rowe, J.: "COMAND" Analysis of the Shuttle Thermal Protection System Tiles. Presented at the Conference on Finite-Element Methods and Technology, Pasadena, California, March 19-20, 1981.

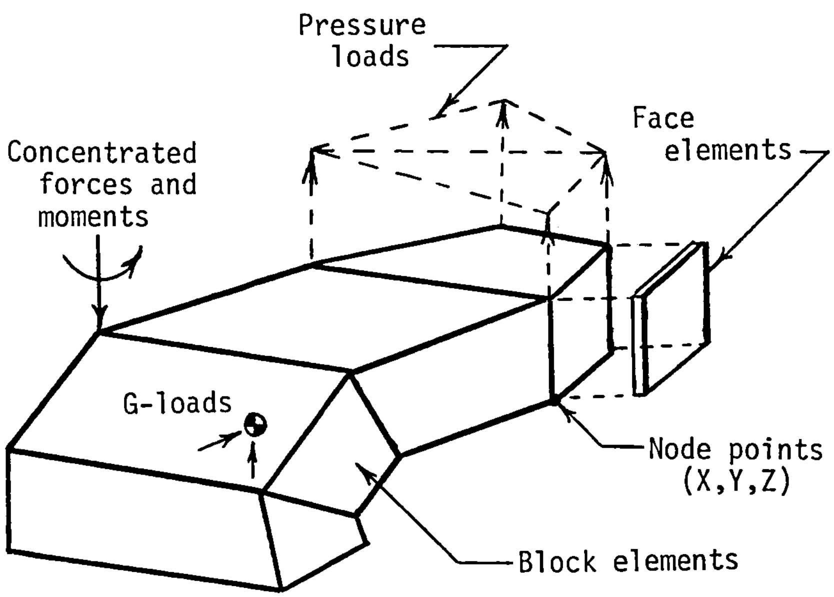
5. Giles, Gary L.; and Haftka, Raphael T.: SPAR Data Handling Utilities. NASA TM 78701, September 1978.

6. FORTRAN Extended Version 4 Reference Manual. Control Data Corporation Pub. No. 60497800, Rev. D, September 1979, pp. 5-8 to 5-12.

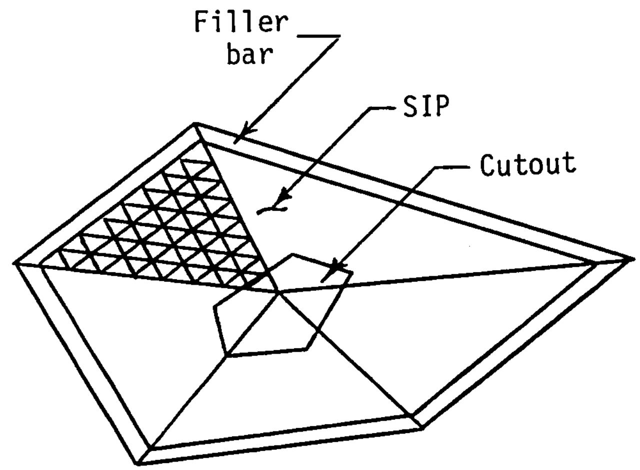
7. Whetstone, W. D.: SPAR Structural Analysis System Reference Manual - System Level 13A. Volume 1 - Program Execution. NASA CR-158970-1, 1978.



(a) TPS arrangement.



(b) Tile modeling.



(c) Nonlinear modeling.

Figure 1.- Analytical model of thermal protection system.

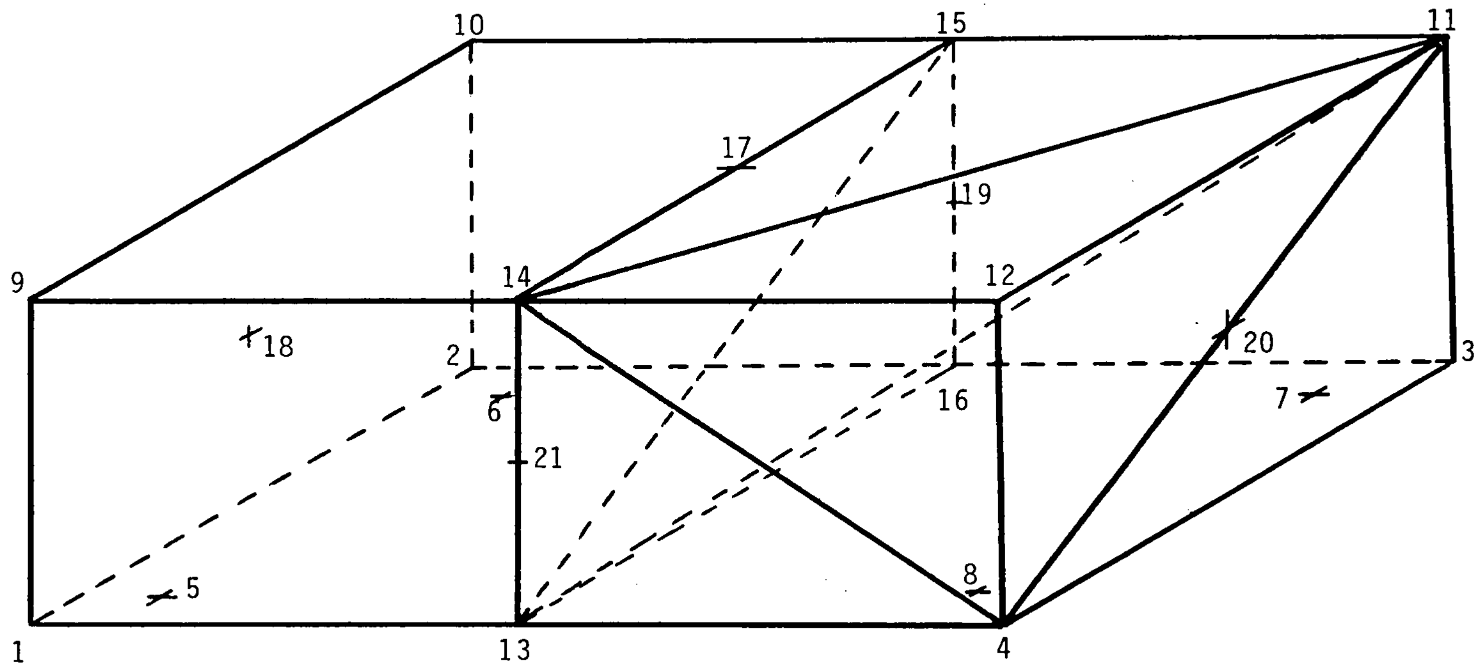


Figure 2.- Geometric model of acreage tile example.

1. Report No. NASA TM-81856	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle COMPUTER PROGRAM FOR NONLINEAR STATIC STRESS ANALYSIS OF SHUTTLE THERMAL PROTECTION SYSTEM - USER'S MANUAL		5. Report Date May 1981
		6. Performing Organization Code 506-89-01-01
7. Author(s) Gary L. Giles and Maria Vallas		8. Performing Organization Report No.
9. Performing Organization Name and Address NASA-Langley Research Center Hampton, VA 23665		10. Work Unit No.
		11. Contract or Grant No.
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546		13. Type of Report and Period Covered Technical Memorandum
		14. Sponsoring Agency Code
15. Supplementary Notes		

16. Abstract User documentation is presented for a computer program which considers the nonlinear properties of the strain isolator pad (SIP) in the static stress analysis of the shuttle thermal protection system (TPS). This program is generalized to handle an arbitrary SIP footprint including cutouts for instrumentation and filler bar. Multiple SIP surfaces can be defined to model tiles in unique locations such as leading edges, intersections, and penetrations. The nonlinearity of the SIP is characterized by experimental stress-displacement data for both normal and shear behavior. Stresses in the SIP are calculated using a Newton iteration procedure to determine the six rigid body displacements of the tile which develop reaction forces in the SIP to equilibrate the externally applied loads. This user documentation gives an overview of the analysis capabilities, a detailed description of required input data and an example to illustrate use of the program.

17. Key Words (Suggested by Author(s)) Thermal Protection System Shuttle TPS Nonlinear Stress Analysis	18. Distribution Statement Unclassified - Unlimited Subject Category 39

19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 58	22. Price A04

N-305

For sale by the National Technical Information Service, Springfield, Virginia 22161

LIBRARY

185