TELINIC 2001220968 555114 208P

NASA/CR—2001-210968

Engine Structural Analysis Software

R.L. McKnight, R.J. Maffeo, S. Schrantz, M.S. Hartle,
G.S. Bechtel, K. Lewis, and M. Ridgway
GE Aircraft Engines, Cincinnati, Ohio

October 2001

The NASA STI Program Office . . . in Profile

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

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.
- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA Access Help Desk at 301–621–0134
- Telephone the NASA Access Help Desk at 301–621–0390
- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

NASA/CR—2001-210968

Engine Structural Analysis Software

R.L. McKnight, R.J. Maffeo, S. Schrantz, M.S. Hartle,
G.S. Bechtel, K. Lewis, and M. Ridgway
GE Aircraft Engines, Cincinnati, Ohio

Prepared under Contract NAS3–26617, Task Order No. 40

National Aeronautics and
Space Administration

Glenn Research Center

October 2001

Note that at the time of research, the NASA Lewis Research Center was undergoing a name change to the NASA John H. Glenn Research Center at Lewis Field. Both names may appear in this report.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at http://gltrs.grc.nasa.gov/GLTRS

TABLE OF CONTENTS

1.0 Introduction

1.1 Task Objectives

The objective of this contract is to enhance Component Specific Modeling (COSMO) software to include geometry recipes for inlets, nozzles, shafts, frames, disks, and ducts. For these components, the software predicts temperature, deformation, stress, and strain histories throughout a complete flight mission.

1.2 Management Plan

The technical effort for this contract is being performed in the Integrated Analysis Methods area of the Advanced Mechanical Technology Operation. The Principal Investigator, charged with the responsibility for ensuring that the technical efforts are completed on schedule, is Dr. Richard McKnight with Mr. Robert Maffeo, Mr. Stephen Schrantz, Mr. Michael Hartle, Mr. Greg Bechtel, Mr. Ken Lewis, and Mr. Mathew Ridgway as the major contributors. The LET Program Manager has responsibility for ensuring that the financial and reporting requirements are met.

1.3 Technical Plan Overview

The objectives of the technical effort in this contract are 1) to develop geometry recipes for nozzles, inlets, disks, frames, shafts, and ducts in finite element form, 2) develop component design tools for nozzles, inlets, disks, frames, shafts, and ducts which utilize these recipes, and 3) develop an integrated design tool which combines the simulations of the nozzles, inlets, disks, frames, shafts, and ducts with the previously developed combustor, turbine blade, and turbine vane models for a total engine representation. These developments will be accomplished in cooperation and in conjunction with the developments in T/BEST, and NPSS. These efforts have been broken down into subtasks A through F, with F covering reporting and deliverables.

1.3.1 Sub Task A Nozzle and Inlet Recipes

Recipes will be developed for nozzles and inlets in conjunction with the designs for the High Speed Civil Transport Engine. These expert programs will be capable of formulating the component geometry as finite element meshes for structural analyses. These recipes will work through neutral files with the T/BEST program which will provide them with their base parameters and loadings.

1.3.2 Sub Task B Disk, Frame, Duct, and Shaft Recipes

Recipes will be developed for disks, frames, ducts, and shafts in conjunction with the designs for the High Speed Civil Transport Engine and the Advanced Subsonic Transport Engine. These expert programs will be capable of formulating the component geometry both as finite element meshes for structural analysis and as NURB geometry for manufacturing. These recipes will work through neutral files with the T/BEST program which will provide them with their base parameters and loadings.

1.3.3 Sub Task C Nozzle and Inlet Component for Multi-phase Design Simulations

Methods will be developed for rule-based recipe stacking for multi-phase design simulations to perform finite element analyses of these components through codes such as CSTEM. These programs will combine the mesh information with the T/BEST loading information to automatically generate the FEA input, run the analysis, and output all or selected portions of the design results. These simulations will be capable of spanning the range from preliminary design needs to final component certification through user controls effecting the fineness/coarseness of the model and simulation level (beam/space shell, static/dynamic, linear/nonlinear).

1.3.4 Sub Task D Disk, Frame, Duct, and Shaft Component for Multi-phase Design Simulations

Methods will be developed for rule-based recipe stacking for multi-phase design simulations to perform finite element analyses of these components through codes such as CSTEM. These programs will combine the mesh information with the T/BEST loading information to automatically generate the FEA input, run the analysis, and output all or selected portions of the design results. These simulations will be capable of spanning the range from preliminary design needs to final component certification through user controls effecting the fineness/coarseness of the model and the simulation level (beam/space shell, static/dynamic, linear/nonlinear).

1.3.5 Sub Task E Integrated Components for Multi-phase Design

The above expert programs will be joined with those previously developed for combustors, turbine blades, and turbine vanes to produce an integrated component design tool in conjunction with T/BEST and NPSS. This code will be capable of running total engine simulations (HSCT, AST, etc.) based on information from and interaction with T/BEST, and NPSS. This code will be developed and built as the individual component codes become available.

1.3.6 Sub Task F Reporting and Deliverables

We shall prepare annual technical reports, deliver enhanced COSMO software, updated COSMO software manual, and quarterly cost summaries. The software will be demonstrated with examples consisting of components from the High Speed Civil Transport Engine and the Advanced Subsonic Transport Engine. The demonstration examples will be recommended by GEAE and approved by the NASA Program Manager.

1.4 Summary of Technical Progress

We have broadened our efforts to encompass components with more complex geometry and loading. Figure 1 shows the previous COSMO recipes. They fall into certain well defined categories. There are the axisymmetric structures such as combustors and disks, the repeating structures, such as frames, and the components with complex surface curves, such as blades, where the surface definition is available as point specific data from aero files. Figure 2 shows the components of the High Speed Civil Transport nozzle system. These components are non-axisymmetric, non-repeating with non-aero complex surface curves, plus attachments such as lugs and clevises. Modeling these required that we develop a new paradigm in order to continue our high automation of the simulation process. Figure 3 illustrates this new paradigm. It involves developing a plan-form. Extruding this plan-form. Then removing elements and gluing on attachments. After this, the loading is applied and the model is spun off to a FEM for analysis.

In addition, we continued to assist the HSCT people with their combustors. As a part of the HSCT-EPM program, we developed, in CSTEM, the ability to contour plot individual layers. We also extended the EZFRAME code to handle composite frame models.

1.4.1 Sub Task A Nozzle and Inlet Recipes

Figure 4 shows the user inputs for the lug or clevis recipe. The only difference between the two being that for the clevis we would continue with extruding and element removal. Figure 5 shows a four lug group generated by the process. Figure 6 shows the recipe plan-form for user defined flaps. The flap geometry is defined by first stacking-up this basic shape into a two dimensional projection of the flap. This basic shape can be thought of as the "building block" of the flap. The next step is to extrude the collection of two dimensional building blocks into the desired three dimensional flap geometry. The last step is to remove the undesired elements. Figure 7 shows the flow of this new process.

Figure 8 is an input file to generate a specific HSCT nozzle flap geometry. There are eight panels specified by L, H1, H2, ALPHA1, ALPHA2, T1, T2, XLOC, and YLOC. This plan-form is then extruded to 13 layers whose thickness are controlled by the dimensions on the last line.

Figure 9 shows the eight panel plan-form generated by the process. Figure 10 is the result of the extrusion process. Figure 11 is the result after element removal and Figure 12 is the finished model with lug attachment.

Additional nozzle geometry's were obtained from the High Speed Civil Transport, HSCT, program. These are contained in Appendix B. Initial recipe strategies have been developed based on these geometry's.

1.4.2 Sub Task B Disk, Frame, Duct, and Shaft Recipes

Frames represent one of the more complex components in an aircraft engine. Frames are typically strutted, static structures that support many of the other major engine components, such as, inlets, bearings, sumps, thrust reversers, augmenters and gear boxes. In addition, the frames provide for the primary load paths to carry maneuver forces and gas loads throughout the engine. Frames are also used to control blade clearances by limiting rotor and stator eccentricity and maintain casing roundness.Most frames are comprised of an inner ring or hub, an outer ring, struts and a bearing cone.

Frames are subjected to a variety of different types of loads. These loads can be typically broken into four major categories: axial loads, radial loads, overturning moments and radial thermal loads. Axial loads usually arise from the rotor force on a thrust bearing or pressure gradients acting on stationary shell elements. Radial loads are typically created by a rotor bearing reaction to a maneuver or vibration condition. Overturning moments most commonly occur in conjunction with radial loads that are axially offset from the plane of the frame. Radial thermal loads are induced by subjecting the inner ring, outer ring and/or struts to differences in thermal growth via temperature gradients or differences in thermal coefficients of expansion. Many of these loads are best represented by a harmonic variation around the circumference of the frame.

Designing an efficient aircraft engine frame involves many aspects of structural analysis. One of the most important attributes that must be predicted for a frame is its deflection as it carries the above loads. This deformation can be characterized by the overall frame stiffness. This stiffness literally dictates the design of the entire engine configuration. It defines the location and number of bearings, the deflection of the engine as a whole, the engine system dynamic critical frequencies and the critical mode shapes.

The frame stiffness is a complex function of all the structural elements that comprise the frame. It can be significantly modified by changing the geometry of the inner and outer rings, the number of struts, the angle at which the struts intersect the rings, the strut cross-sections and the attachment to the bearing cone. It should also be noted that the weight of the frames often represent a significant percentage of the total engine weight.

In order for a weight efficient frame, that satisfies all of the design requirements to be designed in a timely fashion, it is essential that a quick an accurate frame analysis tool be available for the frame designer. The objective of the Frame Analysis subtask of the COSMO project was to develop this tool. This module is a parametrically driven function, where the major parameters correspond to the above mentioned structural elements of a frame, namely inner rings, outer rings and struts. The primary input to this function is a keyword driven file that defines these elements, their number, geometry, material properties, constraints and loadings. All of the major loadings are supported and automatic harmonic variation of the loads is generated. The primary output is a complete 3D finite element model ready to run. Appendix C is the EZFRAMES Users Manual.

1.4.3 Sub Task C Nozzle and Inlet Component for Multi-phase Design Simulations

The model generated by the process of 1.4.1 was spun off to the CSTEM (Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures) code for analysis. The loading imposed was pressure on the bottom, variable temperature, and it was pinned at the lugs and the eighth panel. This model ran in 530 seconds on the HP workstation. Figure 13 is a contour plot of the effective stresses and Figure 14 is a contour plot of the z-stresses. Thus, the recipe process with the new paradigm takes a complex problem and makes it tractable.

1.4.4 Sub Task D Disk, Frame, Duct, and Shaft Component for Multiphase Design Simulations

The GEAE disk design tool, DISKPC (DISK analysis with Plasticity and Creep), has been upgraded and improved and added to the COSMO system. DISKPC performs cyclic inelastic analyses of disks with thermal, centrifugal,and radial mechanical loadings. Disks are analyzed assuming a condition of axisymmetric plane stress. The plasticity analysis is performed by the method of subvolumes. For creep analysis, time-hardening, strain hardening, and life fraction rules are available. Small or large displacement analyses can be run and there is also a restart capability. DISKPC can provide results for non-linear disk analysis in a very small fraction of the time required for a full finite element analysis. A typical DISKPC run takes 20 seconds for a condition similar to a 3 day FEM run. DISKPC post processing generates plots of the results from a selected load case or mission plots of specific data types, such as radial displacement, for various locations in the DISKPC model.

The DISKPC Post Processor was developed and added to COSMO. Another major enhancement was the capability to analyze thermoplasticity with fully reversed plasticity, a necessity when simulating quenching induced residual stresses. Development is continuing on the method to map residual strains from forging finite element models to DISKPC models. This allows residual strains to be input to DISKPC for analysis. Residual stresses can be significant for advanced high temperature, high strength alloys. Disk growth also becomes a significant design consideration in this case. DISKPC provides a fast and simple tool for determining disk growth influenced by residual stresses. Appendix D is the DISKPC User's Manual.

1.4.5 Sub Task E Integrated Components for Multi-phase Design

The COSMO program has been assisting the HSCT program in their efforts to design an advanced technology CMC (Ceramic Matrix Composite) combustor. There have been a succession of designs, each with its' own intricate complexity. The earliest design was a segmented combustor. The composite and optimization capabilities of CSTEM were used on this combustor. Figure 15 shows an axial segment problem definition. The loading include pressure and both convection and radiation thermal loading. The coupled structural/thermal capabilities of CSTEM were used in addition to optimization. Figure 16 shows the results of the optimization and the stress reduction that resulted.

Figure 17 shows the recipe model that was developed for the injector combustor. These models were subsequently analyzed by CSTEM. Figure 18 shows the CSTEM model for the 1997 convoluted combustor. This model was being run for heat transfer.

To assist in evaluating these composite components, the ability has been added to CSTEM to contour plot individual composite layers. Figures 19, 20, & 21 illustrate this capability.

Also, the EZFRAME software has been used to investigate advanced frames. Figures 22 & 23 show a composite frame model with hollow elliptical struts, dual flanges and a soft outer case.

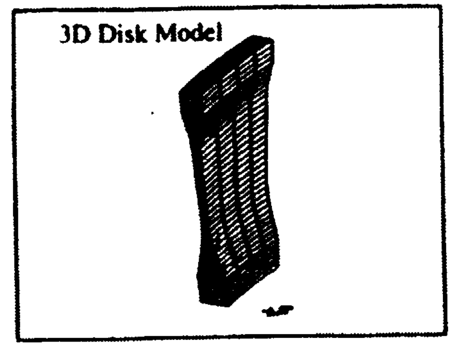
To complement the COSMO work, NYMA Inc. developed a computational procedure that determines temperature dependent metal properties for the Technology Benefit Estimator code, T/BEST. These temperature dependent properties are required by the analysis codes in COSMO to perform structural analyses of the various engine components. T/BEST will act as the data base for these properties. Appendix E covers this capability.

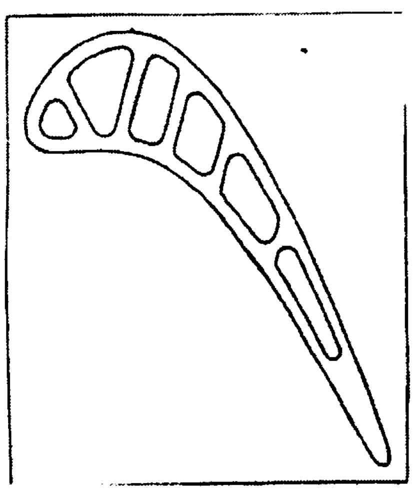
Figure 1 - Existing COSMO Recipes

Axisymmetric (Disks, Combustors)

Repeating Structures (Frames)

Complex Surface Curves (Blades)





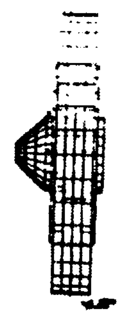


FIGURE 3. EZFRAMES EXAMPLE BYPASS FRAME MODEL

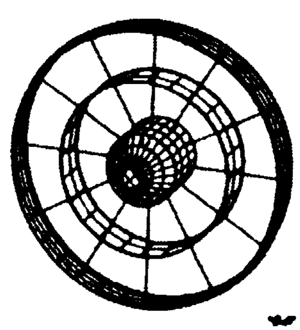


FIGURE 4. EZFRAMES EXAMPLE BYPASS FRAME MODEL

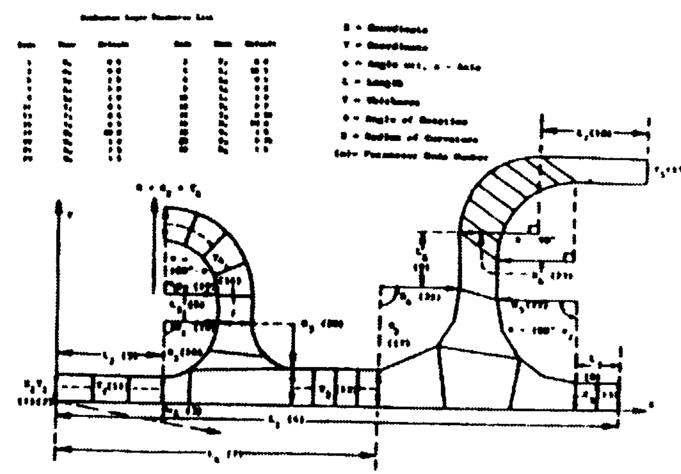


Figure 17. Combustor Liner Parameters

Figure 2 - Nozzle System

Non-Axisymmetric , Non-Repeating , Complex Surface Curves , Plus Attachments

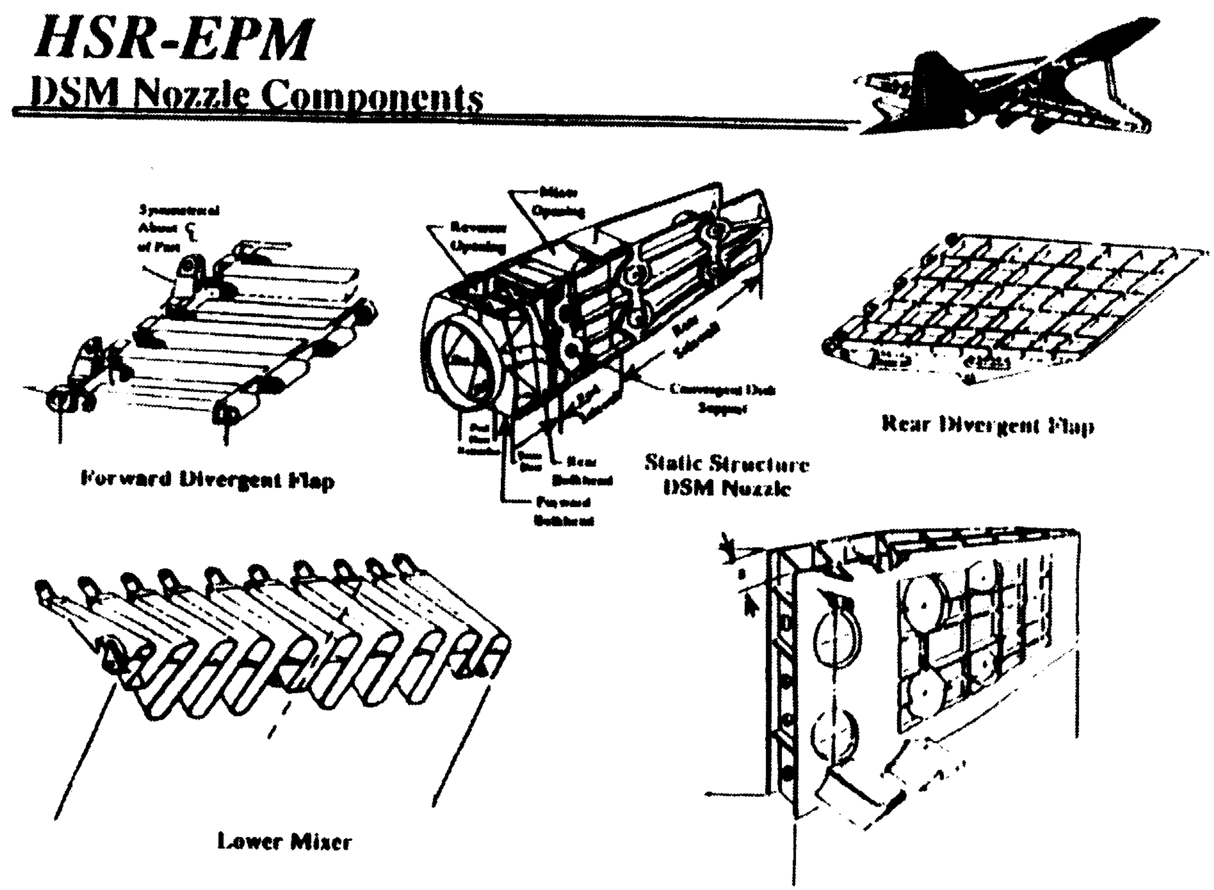
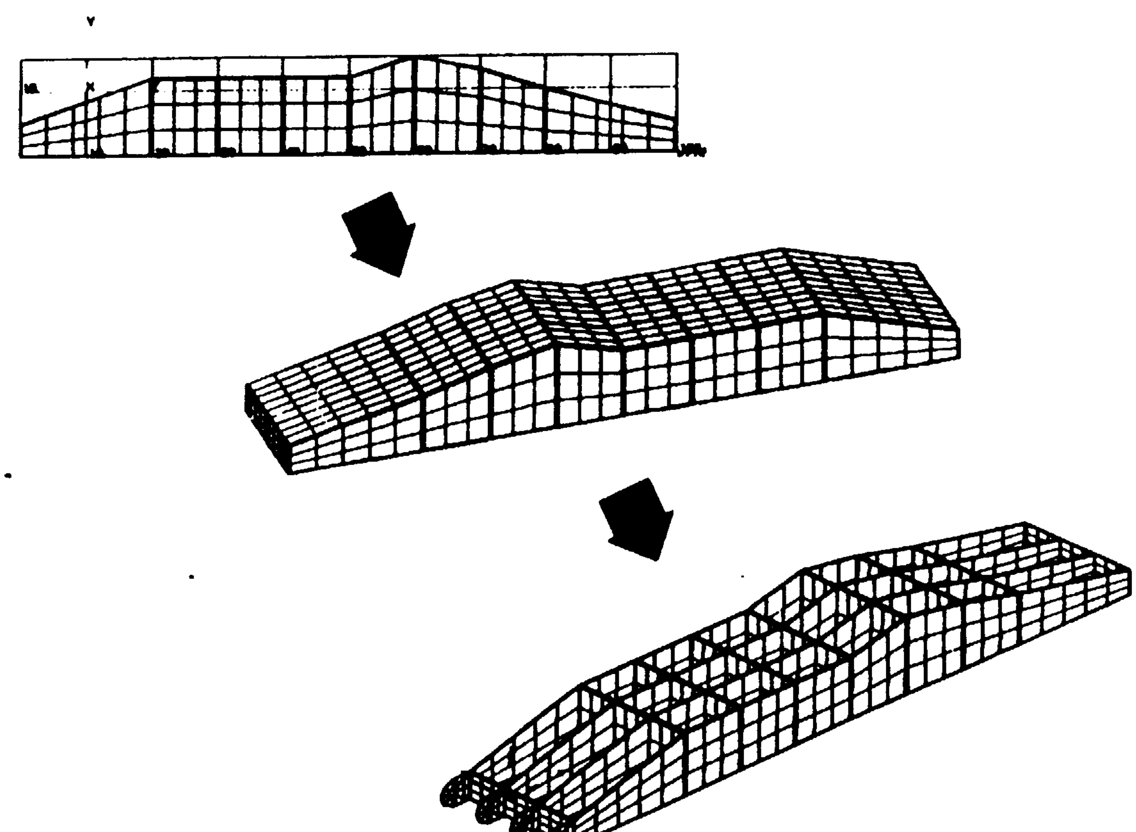


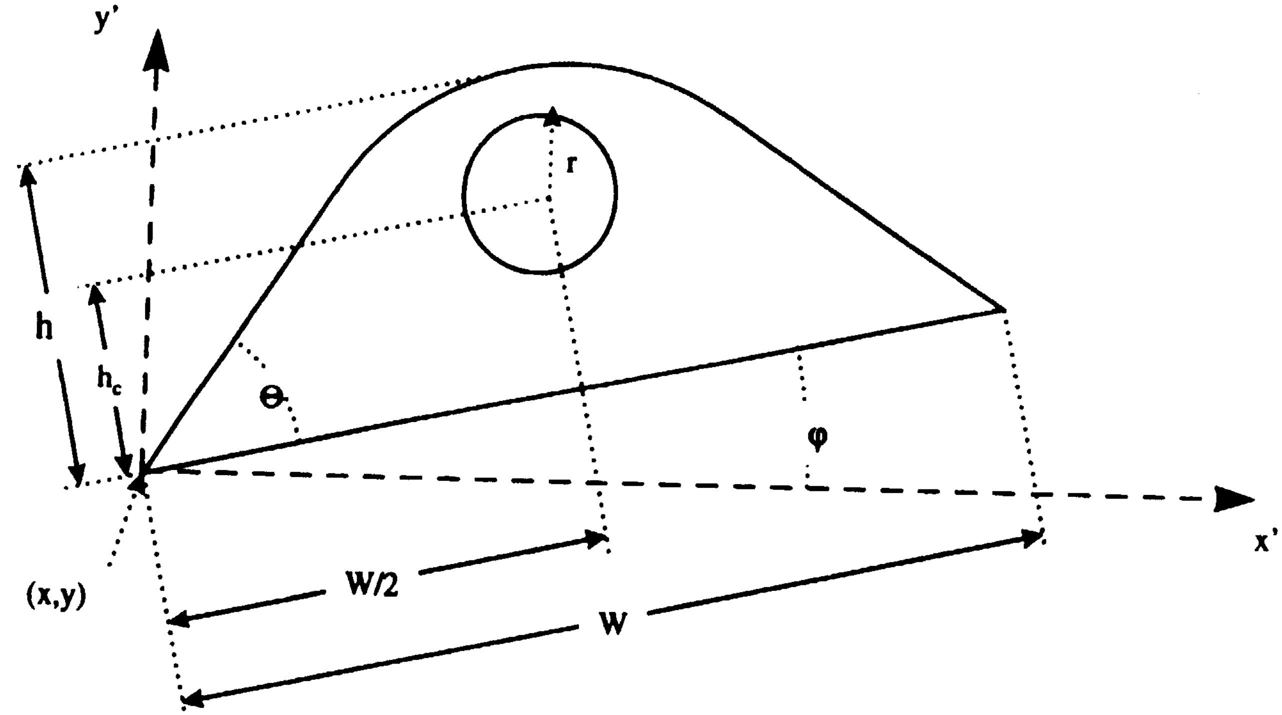
Figure 3 - New Paradigm

Extrude , Remove Elements , and Glue Attachments



(Note: The addition of the lugs.)

Figure 4 - User Inputs for Lug



- Total Height of Lug (h)
- Bottom Width of Lug (W)
- Side Angle (θ)
- Center Height of Hole (h_c)
- Radius of Hole (r)
- Global Offset (x,y)
- Rotation Angle of the Lug (φ)

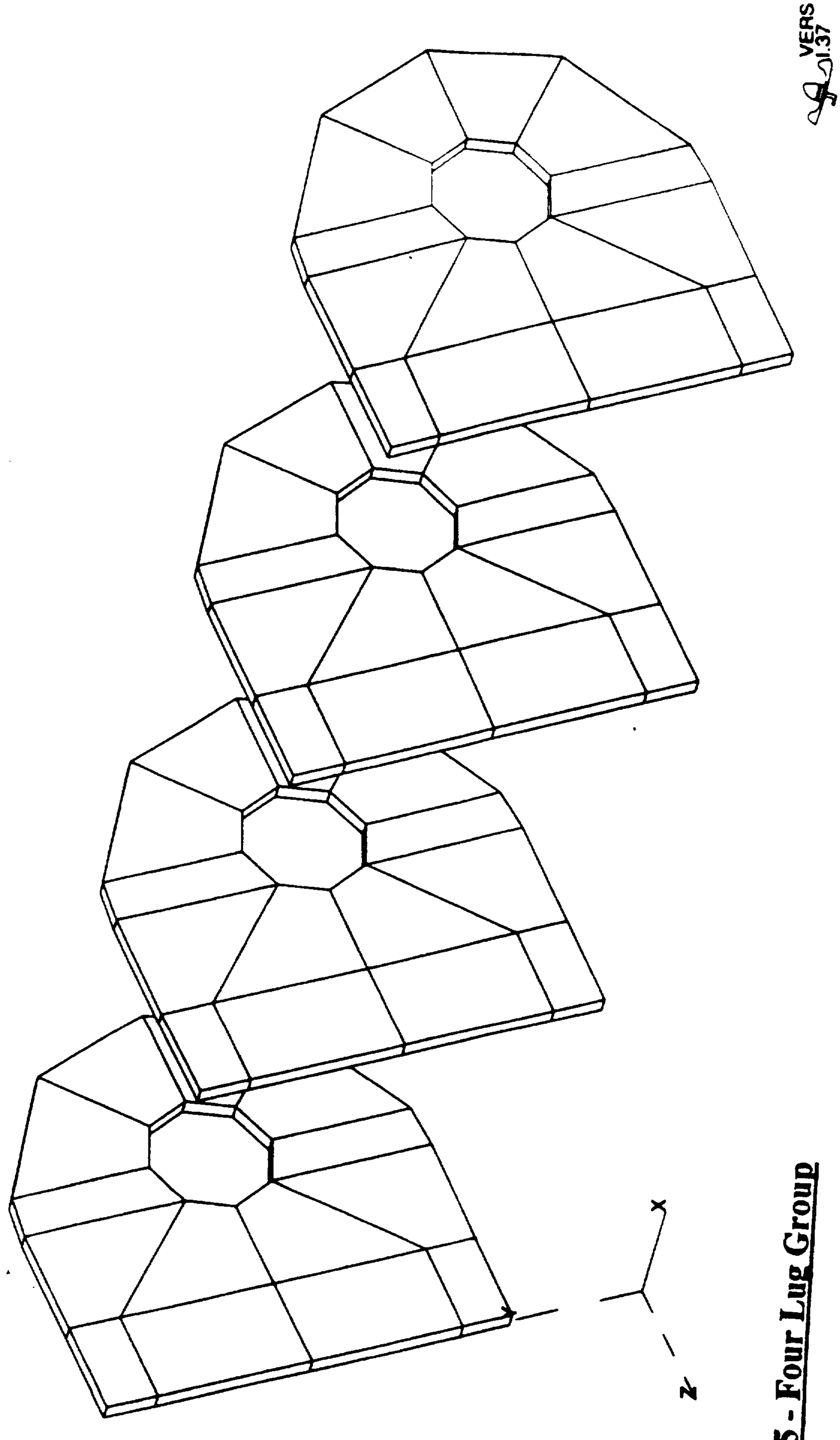
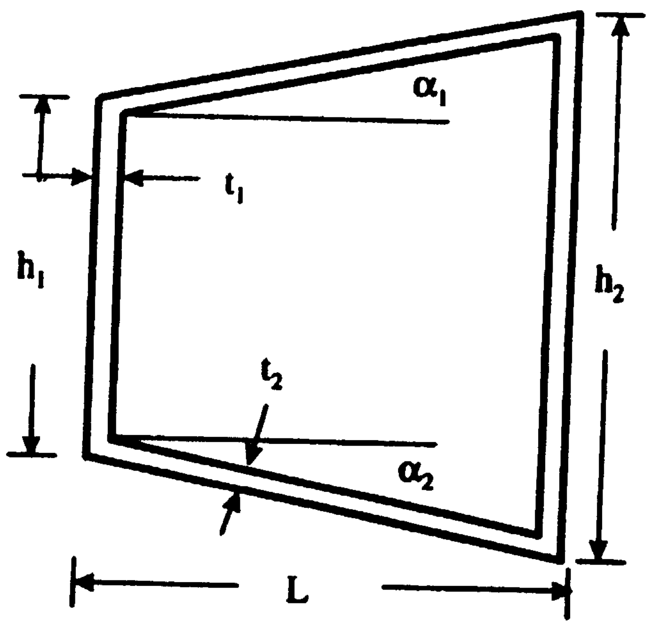


Figure 5 - Four Lug Group

Figure 6 - User Defined Flaps



- Flap geometry is defined by first stacking-up this basic shape into a two dimensional projection of the flap. (See the following page.) This basic shape can be thought of as the "building block" of the flap.
- The next step is to extrude the collection of two dimesional building blocks into the desired three dimensional flap geometry.
- The last step is to remove the undesired elements.
- All the variables shown to the left are the required input (file or keyboard) to successfully execute the program.

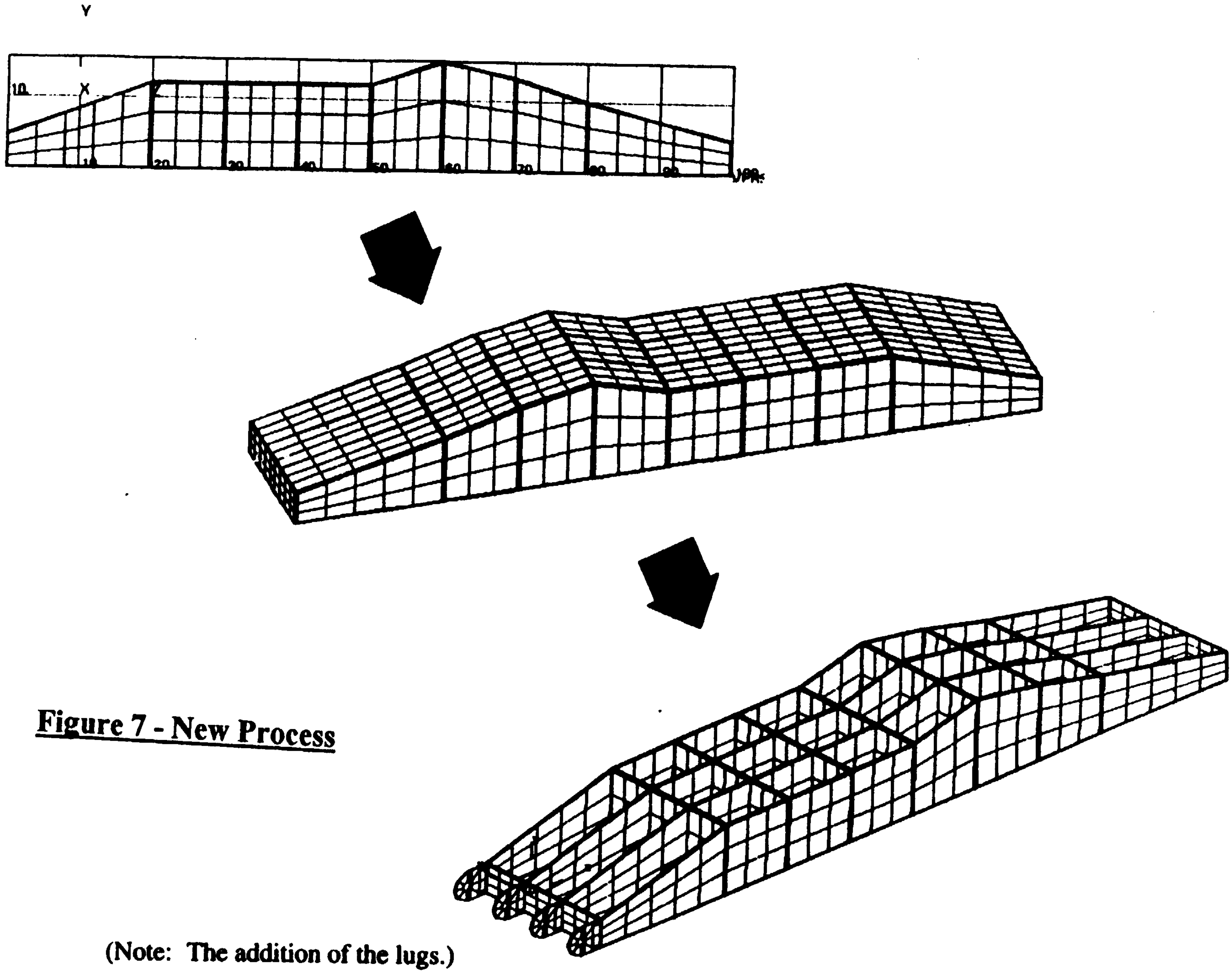


Figure 7 - New Process

(Note: The addition of the lugs.)

Figure 8 - Input File

```
11
PANEL
L H1 H2 ALPHA1 ALPHA2
20,5,12,19.29,0
T1 T2 XLOC YLOC
0.2,0.2,0.,0.
PANEL
L H1 H2 ALPHA1 ALPHA2
10,12,12,0.,0.
T1 T2 XLOC YLOC
0.2,0.2,20.,0.
PANEL
L H1 H2 ALPHA1 ALPHA2
10,12,12,0.,0.
T1 T2 XLOC YLOC
0.2,0.2,30.,0.
PANEL
L H1 H2 ALPHA1 ALPHA2
10,12,12,0.,0.
T1 T2 XLOC YLOC
0.2,0.2,40.,0.
PANEL
L H1 H2 ALPHA1 ALPHA2
10,12,15,16.7,0.
T1 T2 XLOC YLOC
0.2,0.2,50.,0.
PANEL
L H1 H2 ALPHA1 ALPHA2
10,15,13,-11.31,0.
T1 T2 XLOC YLOC
0.2,0.2,60.,0.
PANEL
L H1 H2 ALPHA1 ALPHA2
10,13,10,-16.7,0.
T1 T2 XLOC YLOC
0.2,0.2,70.,0.
PANEL
L H1 H2 ALPHA1 ALPHA2
20,10,5,-14.04,0.
T1 T2 XLOC YLOC
0.2,0.2,80.,0.
wuif
ruif
extr
13
0.2,2.2,4.2,6.2,6.4,8.4,10.4,12.4,12.6,14.6,16.6,18.6,18.8
```

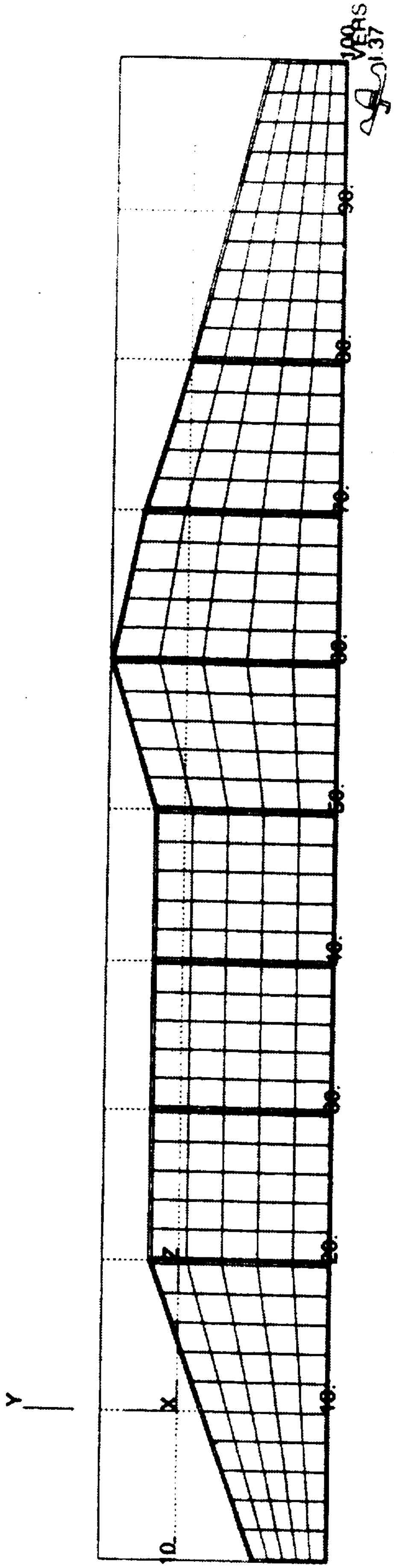


Figure 9 - Eight Flap Section

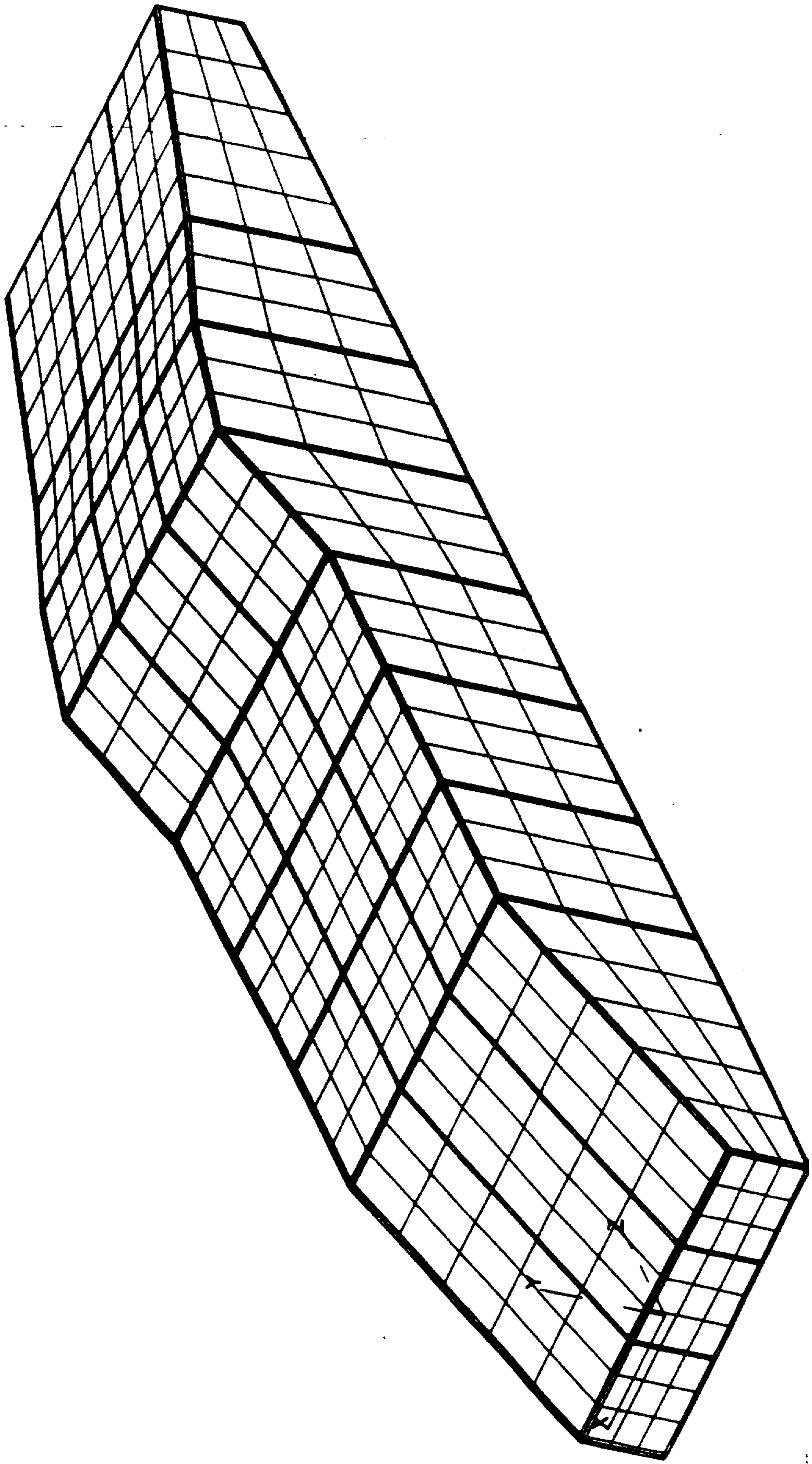


Figure 10 - Extruded Flaps

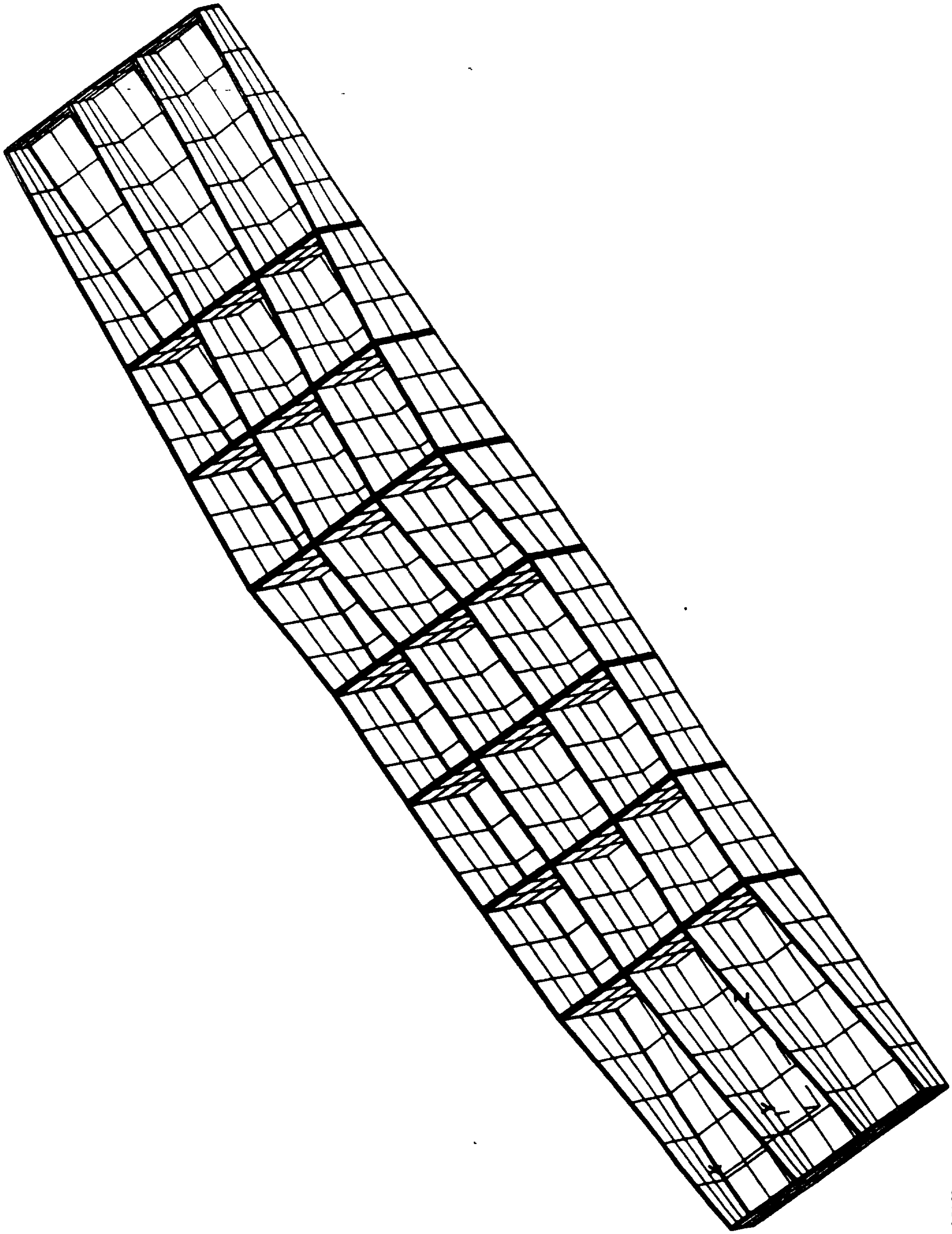
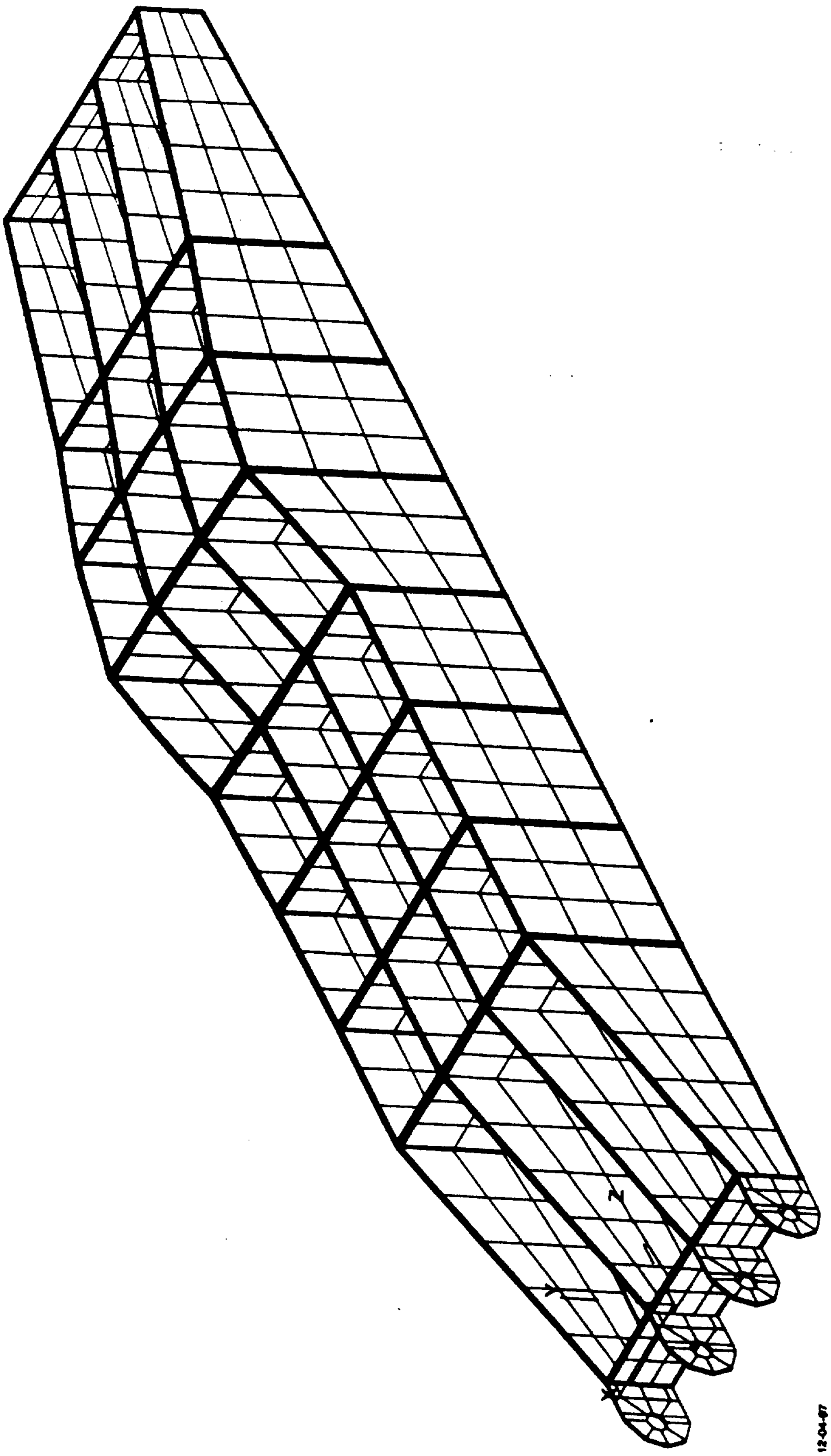


Figure 11 - Element Removal



12-04-97

Figure 12 - Lug Attachment

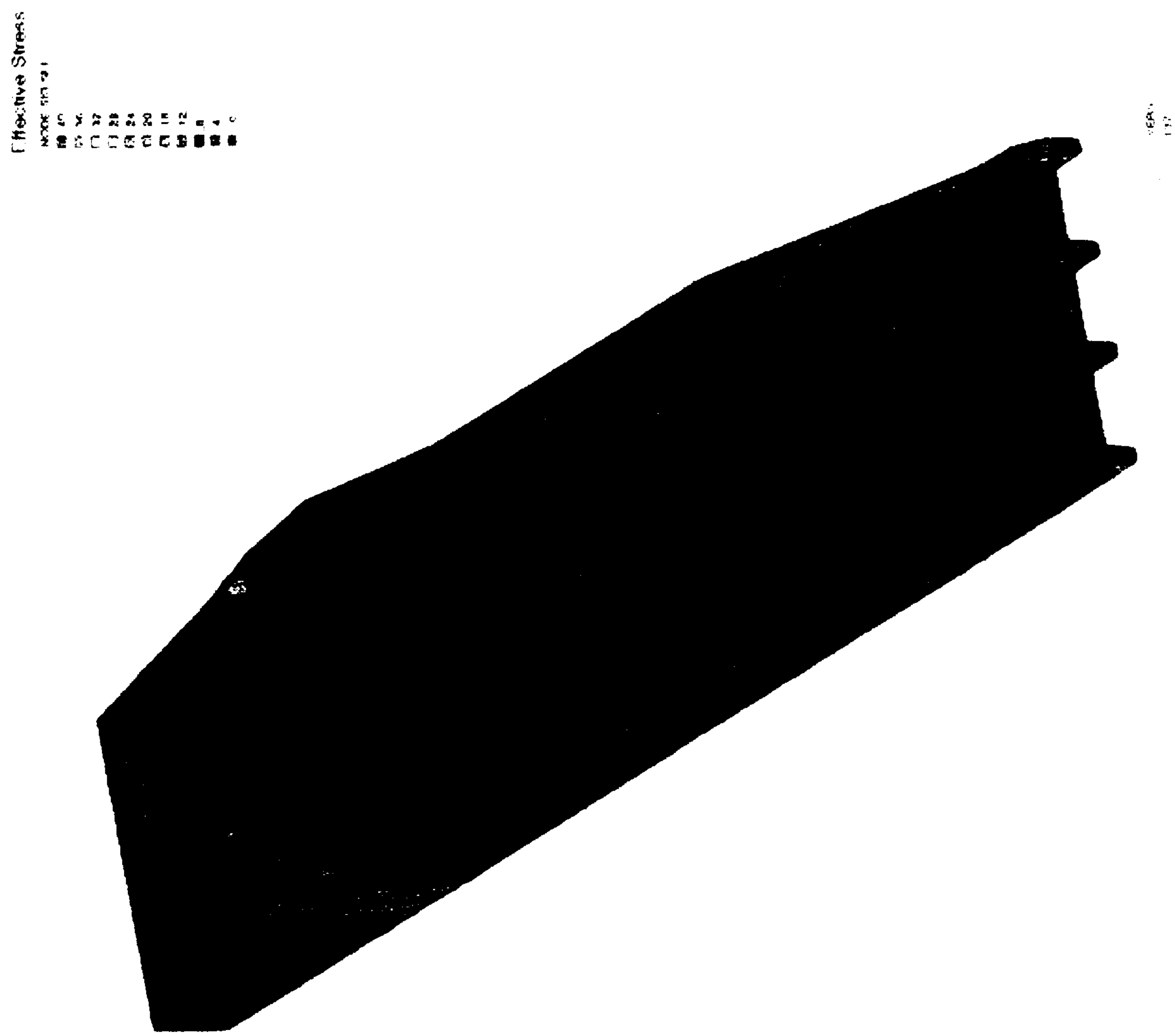


Figure 13
CSTEM Effective Stress Contours

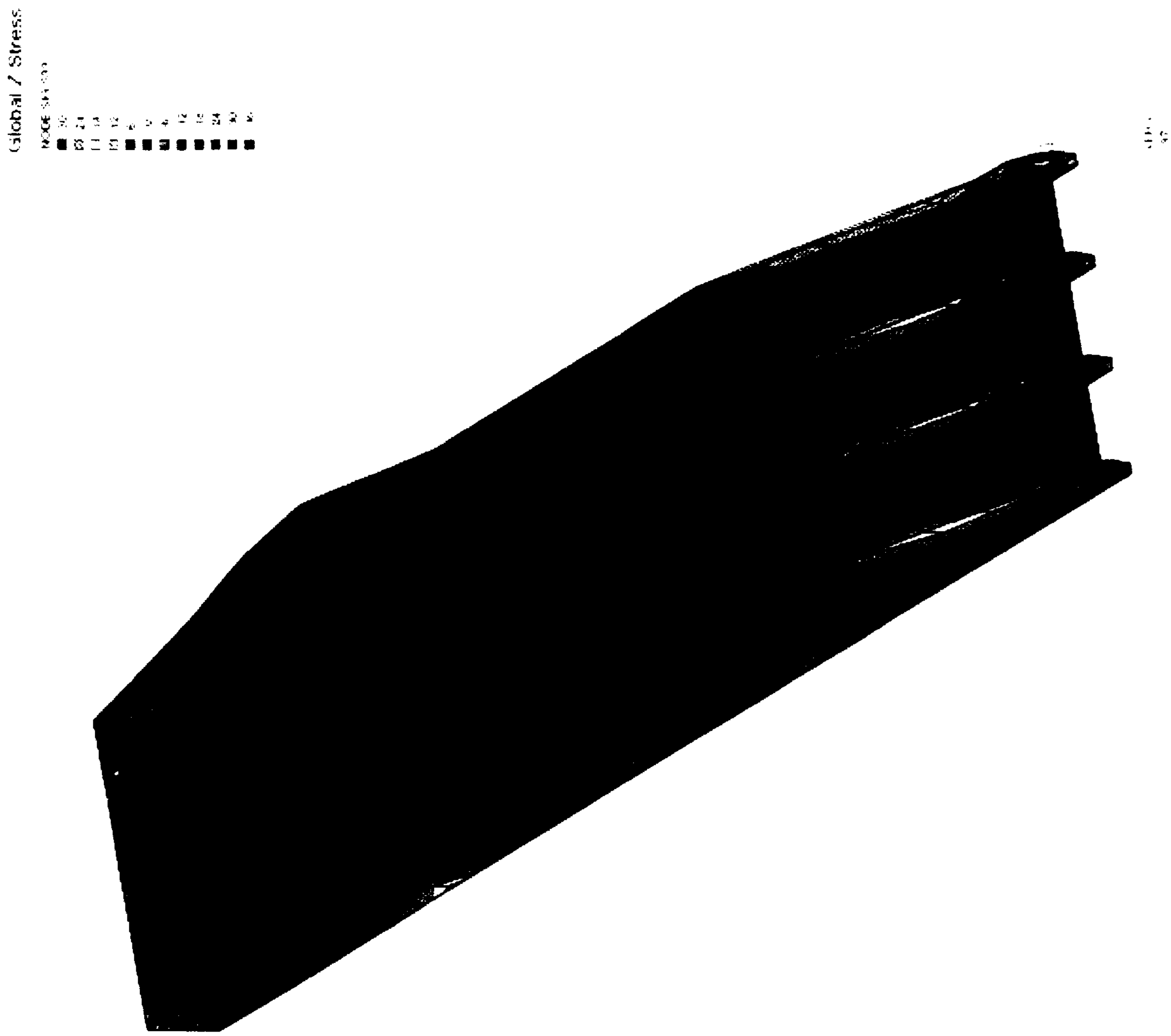


FIGURE 14
CSTEM Z-STRESS CONTOURS

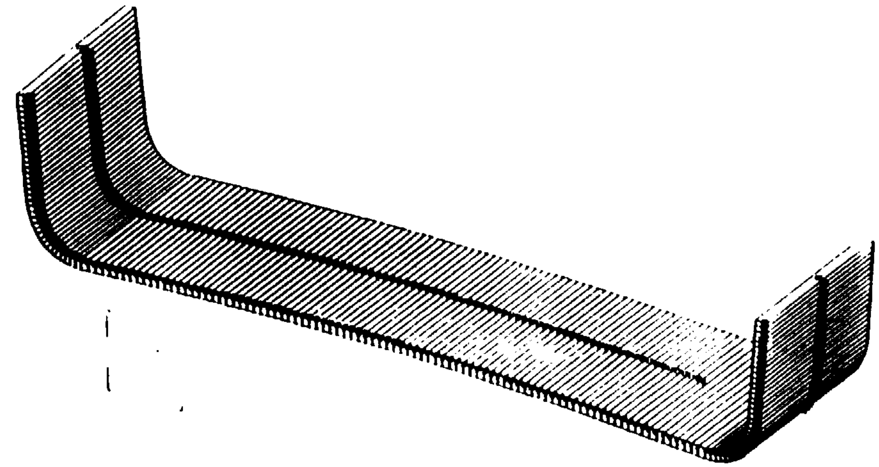
Figure 15 - Axial Segment - Problem Definition

F.E.Model: 20 noded bricks, 160 elements

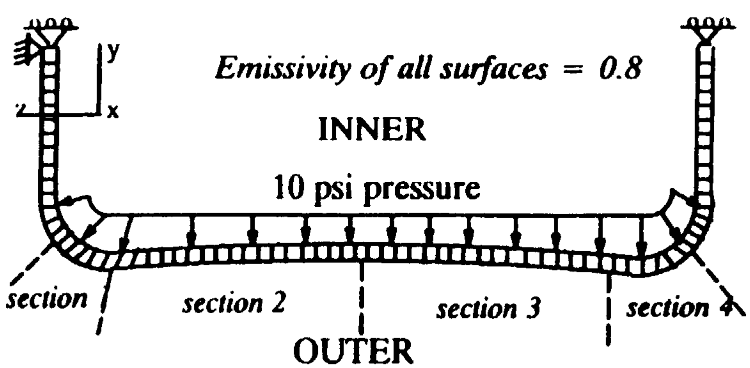
Material: SiC/SiC Ceramic Matrix Composites

Ply Orientation: 1 generation set with 6 layers
[0, 0, 0, 0, 0, 0]

Loading: Thermal boundary conditions on the inner and outer surfaces that vary along the length and 10 psi pressure on inner surface.



$$\left[{}^{t}\mathbf{K}^{k} + {}^{t}\mathbf{K}^{c} + {}^{t}\mathbf{K}^{r} \right] \Delta\theta^{(i)} = {}^{t+\Delta t}\mathbf{Q} + {}^{t+\Delta t}\mathbf{Q}_{c}^{(i-1)} + {}^{t+\Delta t}\mathbf{Q}_{r}^{(i-1)} - {}^{t+\Delta t}\mathbf{Q}_{k}^{(i-1)}$$



INNER SURFACE	Section 1	Section 2	Section 3	Section 4
Convection coefficient (BTU/hr-ft-°F)	292	331	289	265
Convection temp (°F)	1260	1363	1454	1488
Radiation temp (°F)	1350	1436	1478	1558
OUTER SURFACE				
Convection coefficient (BTU/hr-ft-°F)	144	95	91	96
Convection temp (°F)	3235	3235	3235	3235
Radiation temp (°F)	3235	3235	3235	3235

Heat transfer solution as input loads for Structural solution

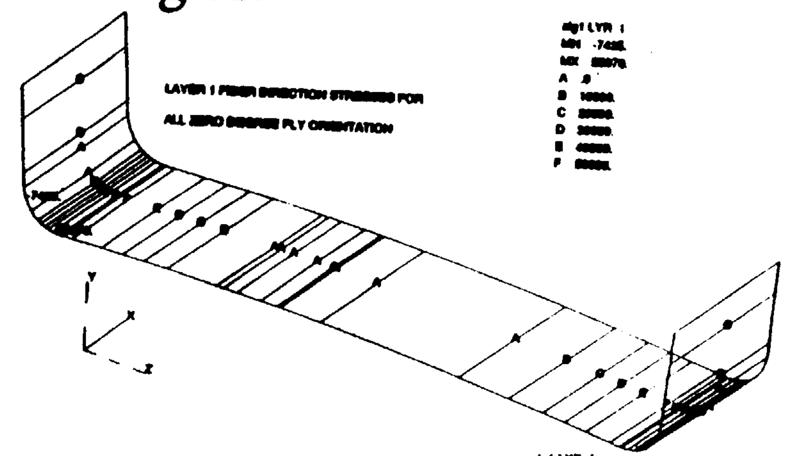
Figure 16 - Axial Segment - Optimization

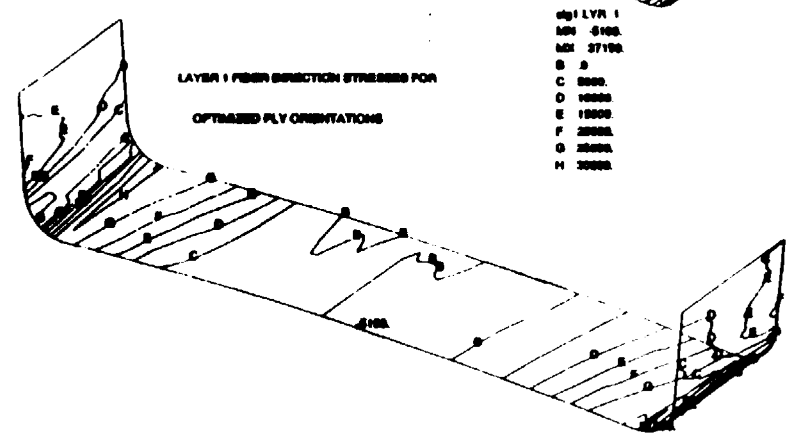
Goal: To find the ply orientations that give the least thermal stresses.

Objective: Minimize: Stress in fiber direction ($Sigma_{11}$)

Constraints: In–plane transverse stresses ($Sigma_{22}$) $\leq$ $Sigma_{11}$
& Bounds on ply orientation angles.

Data	Initial	Final
θ_1 degrees	0	45.
θ_2	0	40.
θ_3	0	33.
θ_4	0	–30.
θ_5	0	–18.
θ_6	0	–9.
Max. $Sigma_{11}$	54,577 psi	39,553 psi
Max. $Sigma_{22}$	24,578 psi	38,954 psi





27% Reduction in Thermal Stresses

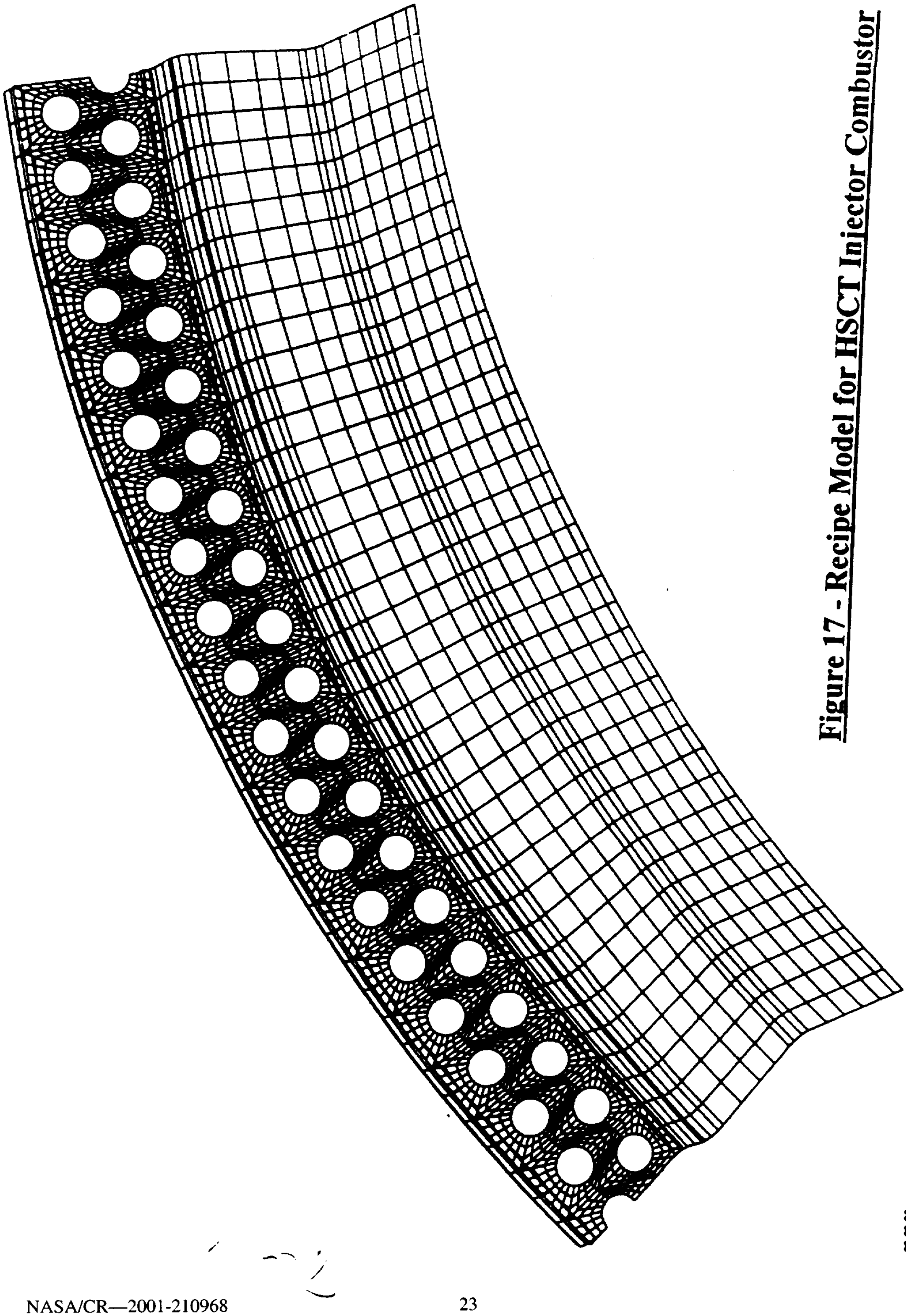
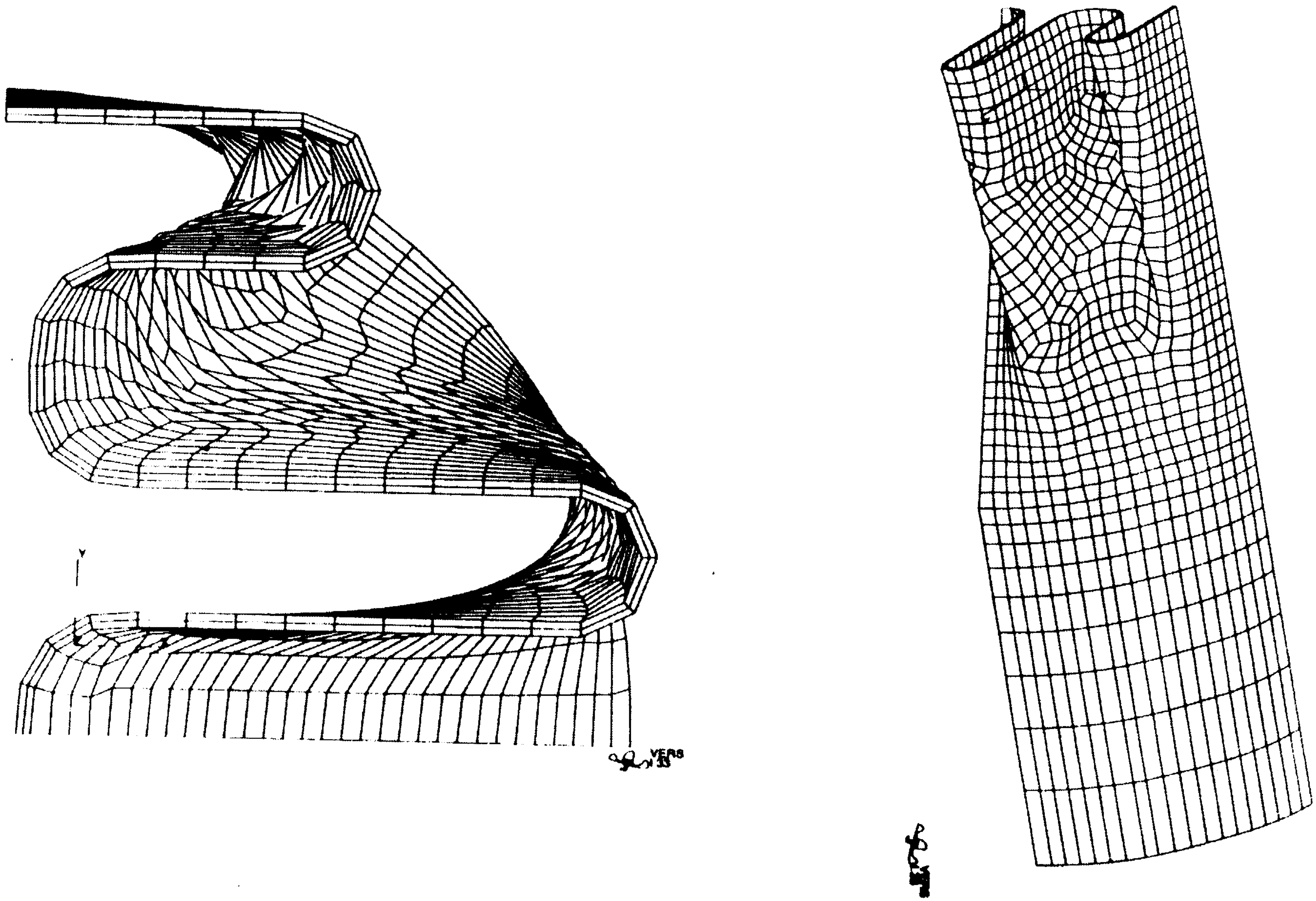


Figure 17 - Recipe Model for HSCT Injector Combustor

Figure 18 - CSTEM Model for 1997 HSCT Combustor



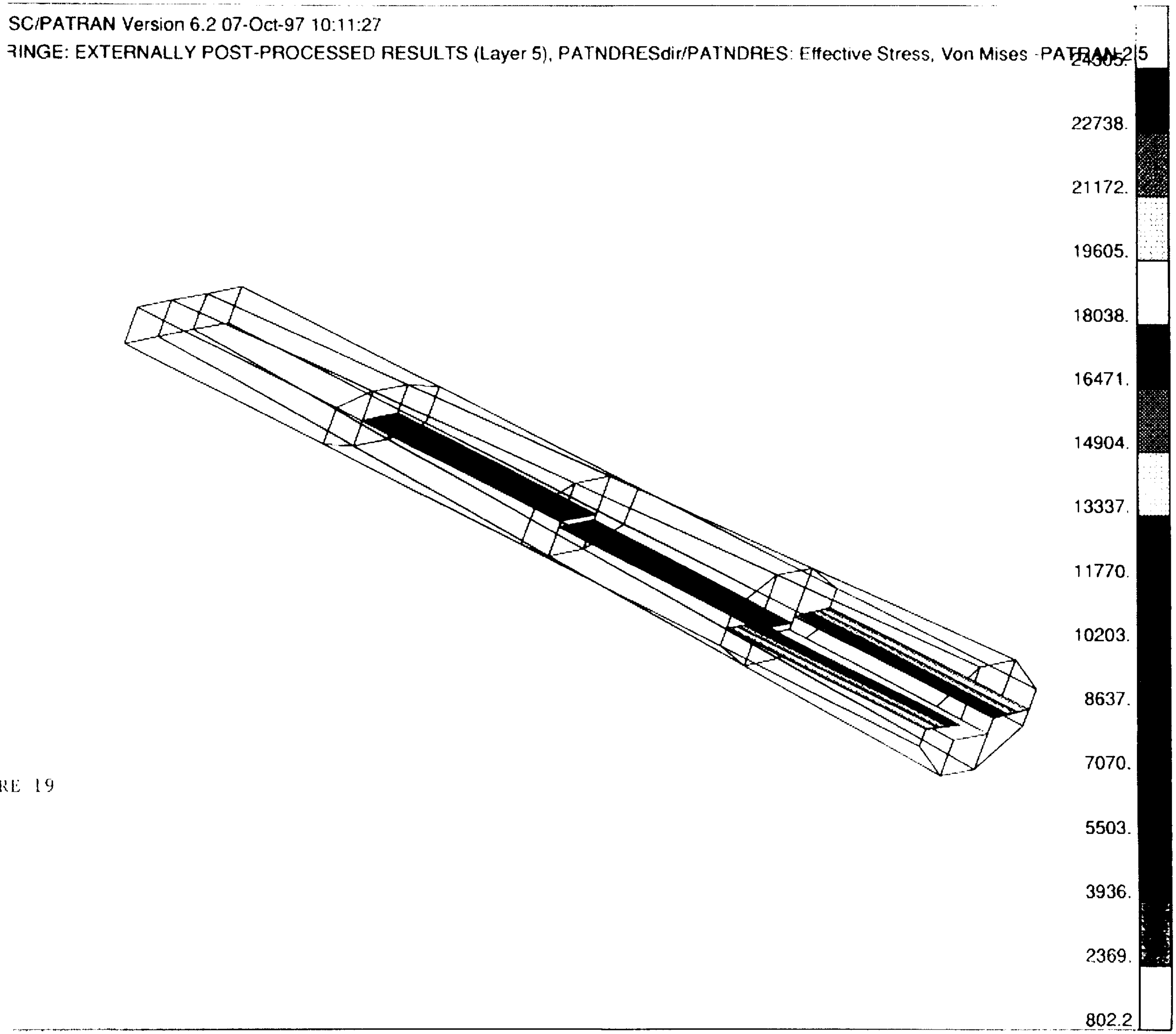

SC/PATRAN Version 6.2 07-Oct-97 10:11:27
RINGE: EXTERNALLY POST-PROCESSED RESULTS (Layer 5), PATNDRESdir/PATNDRES: Effective Stress, Von Mises -PATRAN-2.5
22738.
21172.
19605.
18038.
16471.
14904.
13337.
11770.
10203.
8637.
7070.
5503.
3936.
2369.
802.2

FIGURE 19

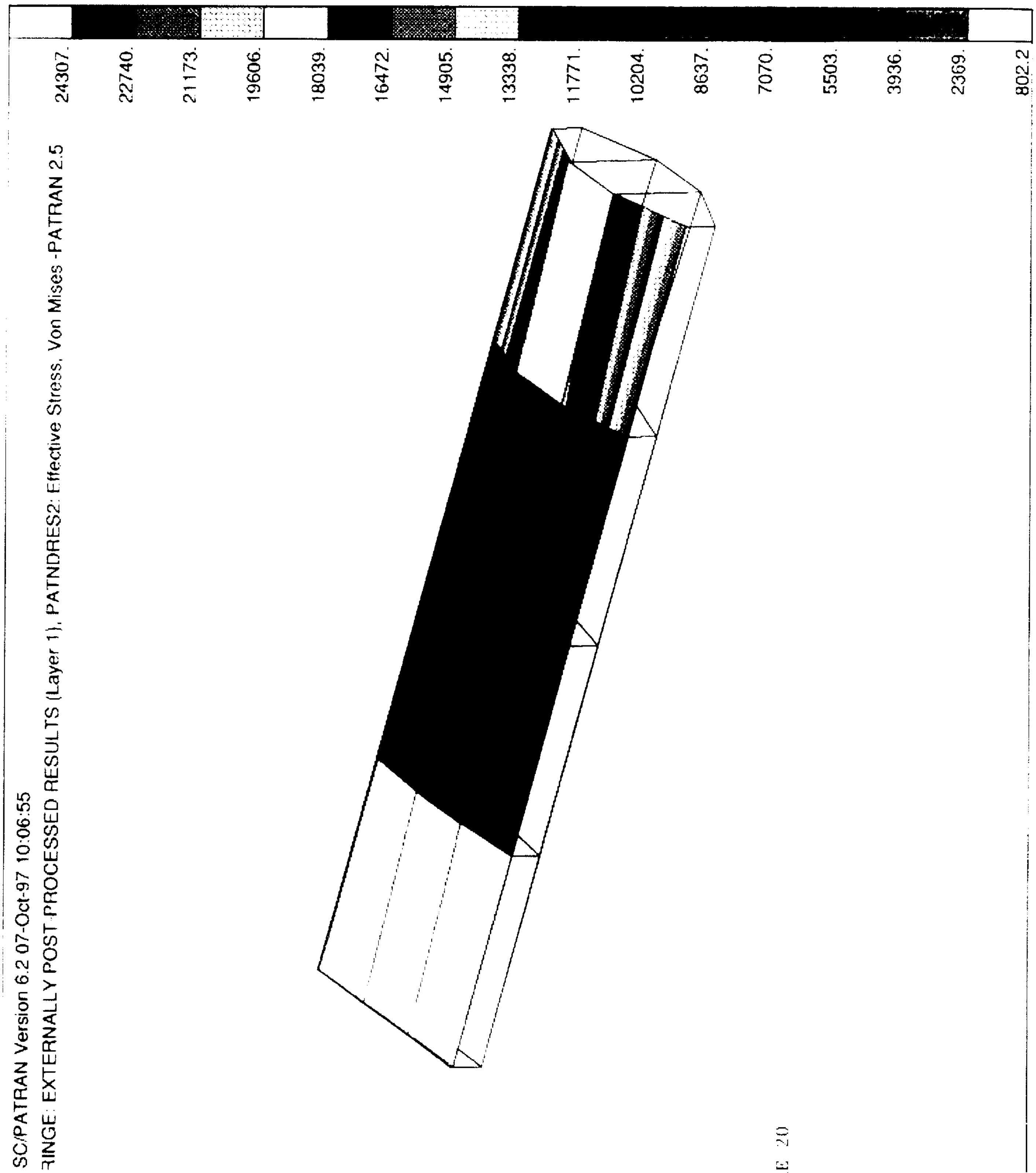

SC/PATRAN Version 6.2 07-Oct-97 10:06:55
RINGE: EXTERNALLY POST-PROCESSED RESULTS (Layer 1), PATNDRES2: Effective Stress, Von Mises -PATRAN 2.5
24307.
22740.
21173.
19606.
18039.
16472.
14905.
13338.
11771.
10204.
8637.
7070.
5503.
3936.
2369.
802.2

FIGURE 20

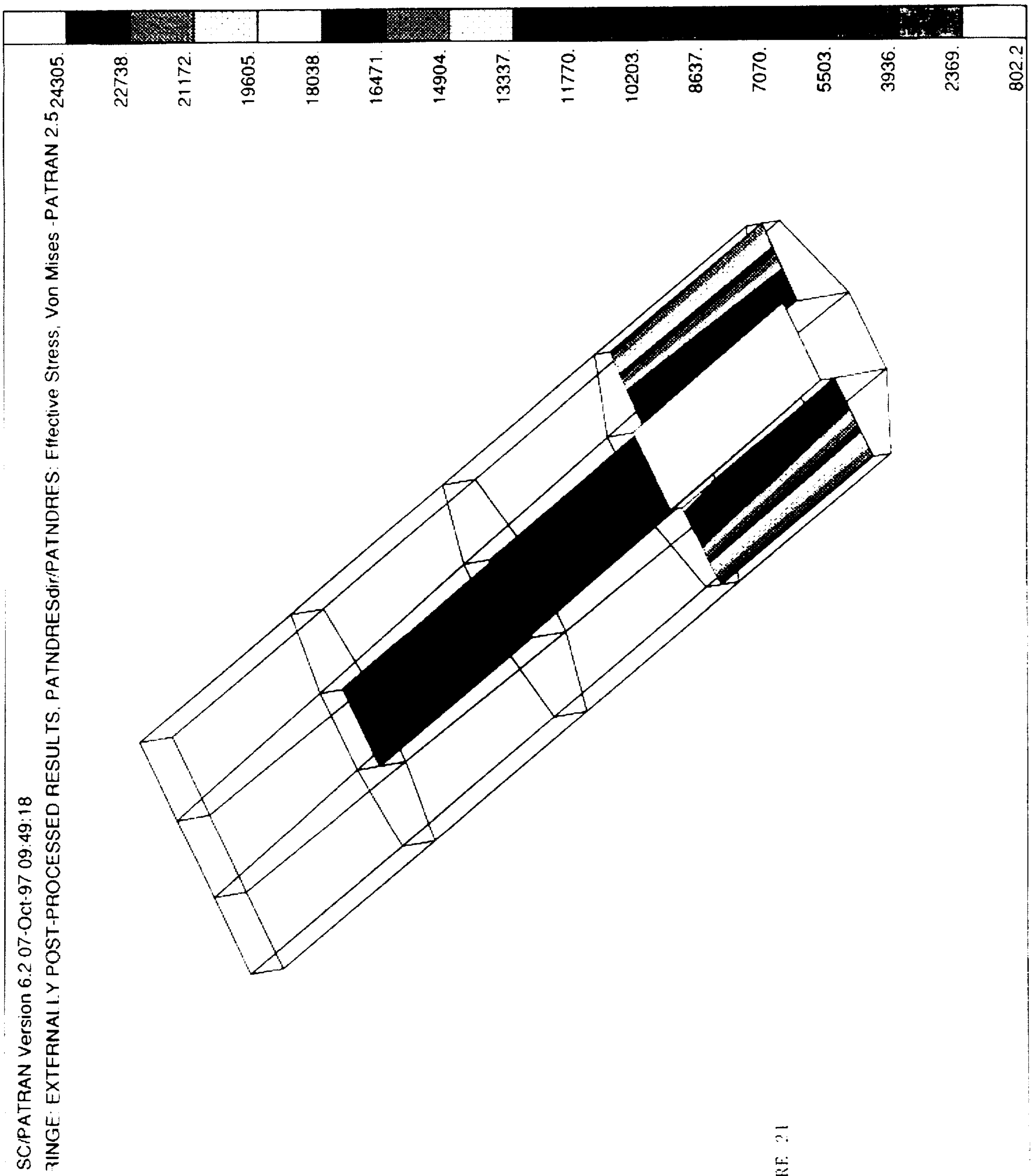

SC/PATRAN Version 6.2 07-Oct-97 09:49:18
RINGE: EXTERNALLY POST-PROCESSED RESULTS, PATNDRESdir/PATNDRES: Effective Stress, Von Mises -PATRAN 2.5
24305.
22738.
21172.
19605.
18038.
16471.
14904.
13337.
11770.
10203.
8637.
7070.
5503.
3936.
2369.
802.2

FIGURE 21

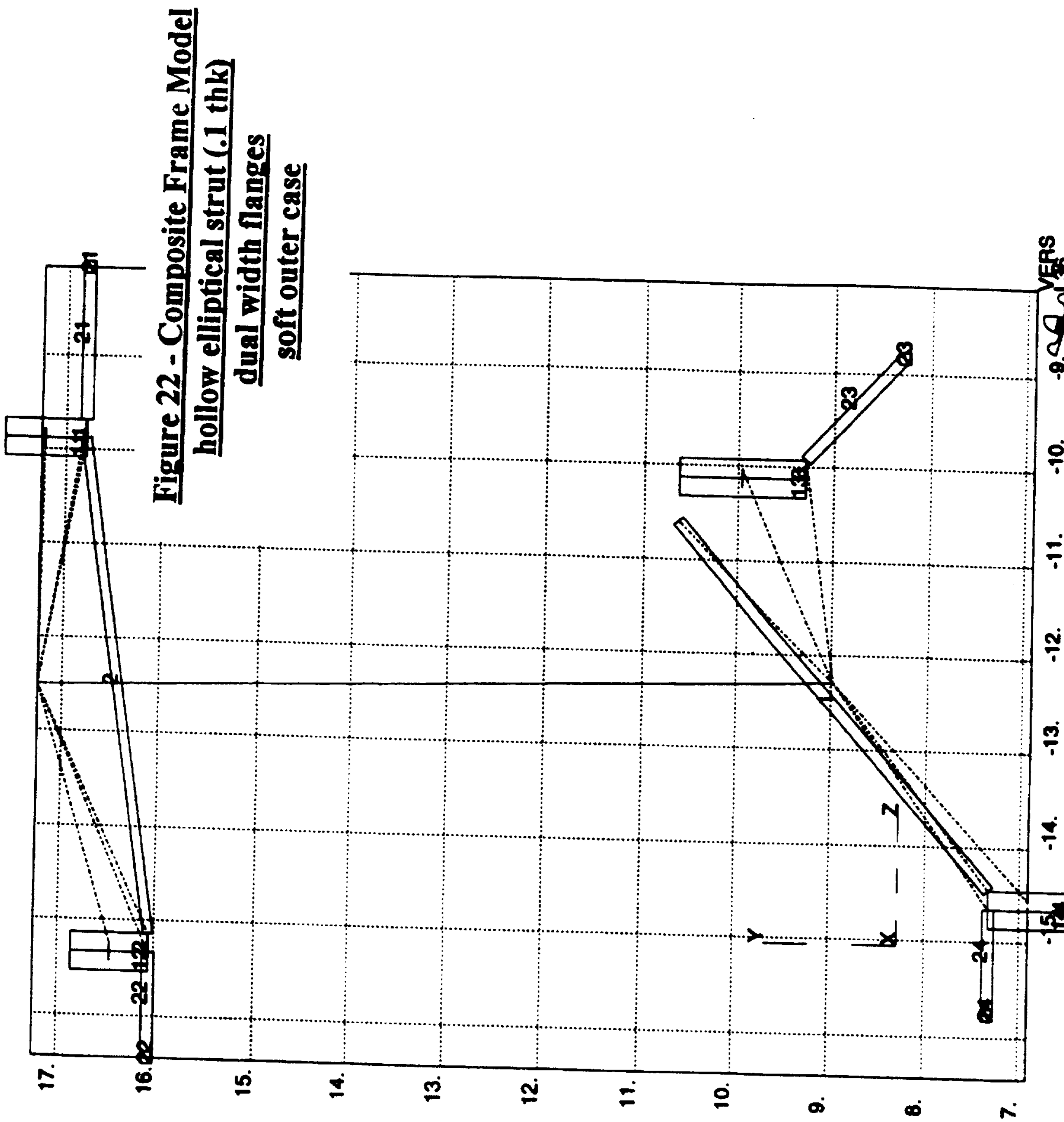


Figure 22 - Composite Frame Model
hollow elliptical strut (.1 thk)
dual width flanges
soft outer case

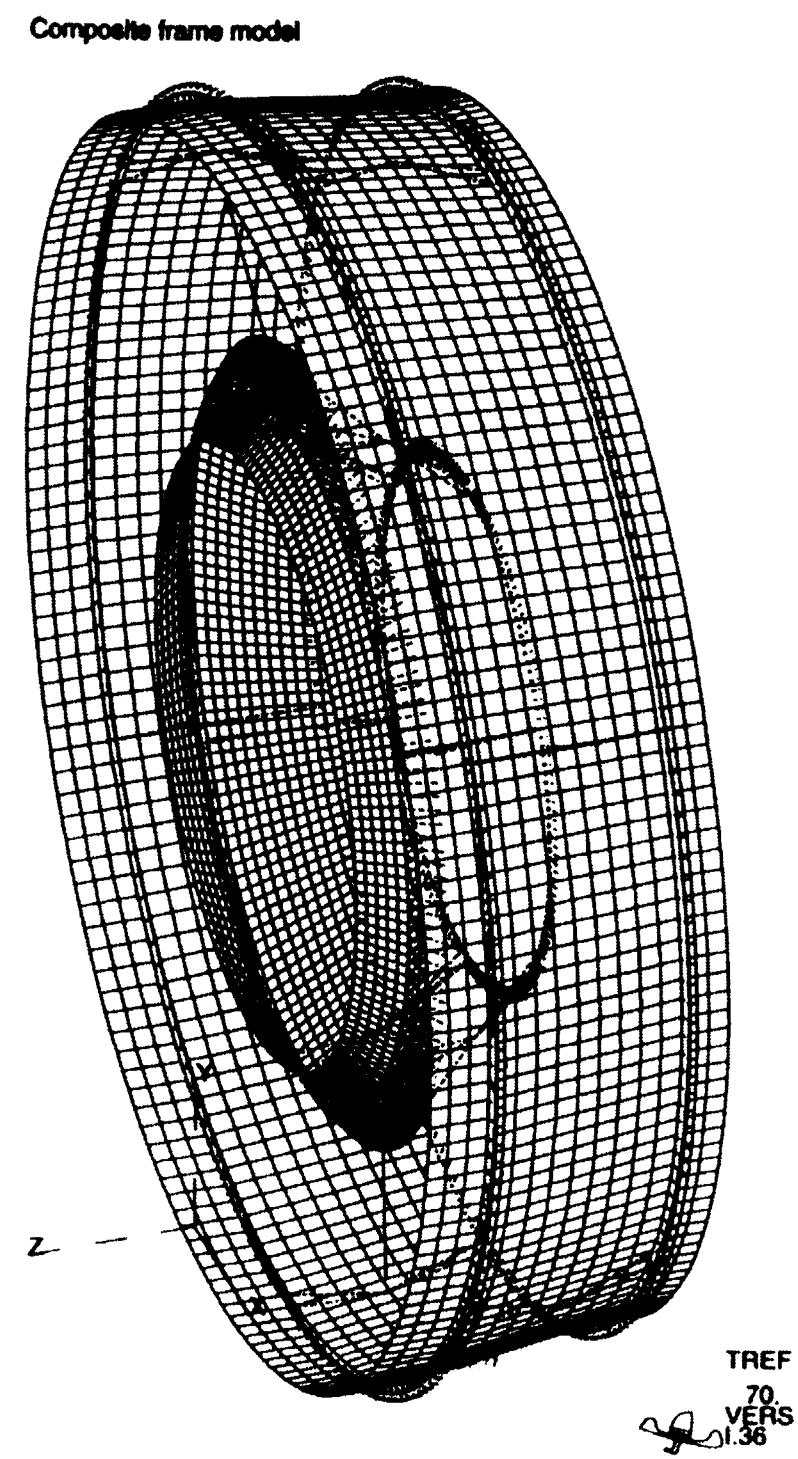


Figure 23 - Composite Frame Model
hollow elliptical struts
dual flanges (Ti - composite)
soft outer case

APPENDIX A

COSMO USER'S MANUAL

1. COSMO SYSTEM

1.1. Introduction

The Component Specific Structural Modeling (COSMO) System is a product of the GE Aircraft Engines' Product Definition and Analysis Methodologies Sub-Section and is intended to consolidate and streamline a number of the functions involved in structural analysis of aircraft engine components for NASA. This manual is for Version I.01 of COSMO which was released in the second quarter of 1994. This version of COSMO is an expansion of the original COSMO software, documented in the final report for NAS3-23687 (Component-Specific Modeling).

1.2. Essential Definitions and Jargon

As with any computer program, there are a number of definitions and abbreviations which you need to understand in order to use COSMO or this manual. The following is a list of abbreviations, program names, etc. which you may encounter in using COSMO.

ANSYS	A general purpose finite element analysis program supplied by Swanson Analysis Systems.
ASCII	Abbreviation for American Standard Code for Information Interchange. This is a method of representing character data as a binary number. This format is used on most computer systems for storing plain, old listable files. Therefore, these are usually called "ASCII Files."
CSTEM	A finite element analysis program developed by GEAE under NASA contract. CSTEM was specifically developed for analyzing composite structures.
IGES	Abbreviation for International Graphics Exchange Specification, a file format for geometric data. This format is supported by many commercial drafting systems and computer programs.
NASTRAN	A general purpose finite element analysis program supplied by MacNeil Schwindler Co. (MSC).
Neutral File	An input format used to define component data and mission data for COSMO (T/BEST).
PATRAN	A general purpose finite element pre and post processing program supplied by PDA Engineering.
RDB	Abbreviation for Random Data Base. This is a random, system dependent file created from a UIF.
SIESTA	GEAE finite element pre and post processing program. Performs many supporting functions in COSMO for model generation.

T/BEST	NASA Technology Benefit Estimator.
UIF	Abbreviation for Unified Input File. This is an ASCII file used for defining input data in COSMO. The format of this file is described in Section 2 of the SIESTA manual.
UOF	Abbreviation for Unified Input File. This is an ASCII file used for defining output data in COSMO. The format of this file is described in Section 3 of the SIESTA manual.

1.3. Current Functions and Function Summaries

A summary of all functions currently available in COSMO, are listed in Section 1.4. A complete listing of the current menu is available at any time from COSMO using the MENU keyword. For more information on a specific function, you should consult the detailed descriptions in Section 1.6. The first page of each function description is a summary which may be useful in determining if a function is suitable for your particular need. The format of this summary with examples is shown below.

> **SAMPLE FUNCTION SUMMARY**
>
> FUNCTION:
>
> **This is the "name" of the function.**
> **EXAMPLE: Create Random Data Base from Unified Input File**
>
> SUB-MENU LOCATION:
>
> **This is the Sub-Menu and Function number for this function.**
> **EXAMPLE: Data Base (Sub-Menu 1, Function 1 or UIFREAD)**
>
> PURPOSE:
>
> **This is a short description of the capabilities and use of this function.**
> **EXAMPLE: This function reads a Unified Input File (UIF), checks the syntax and writes a Random Data Base (RDB).**
>
> INPUT FILES:
>
> **This is a list of any required or optional input files, as well as the default file code.**
> **EXAMPLE: UIF (31)**
>
> OUTPUT FILES:
>
> **This is a list of any output files produced by the function along with the file code.**
> **EXAMPLE: RDB (37)**
>
> REQUIRED USER INPUT:
>
> **This is a brief summary of required interactive input.**
> **EXAMPLE: None, unless terminal input is requested in the UIF.**
>
> COMMENTS:
>
> **This is usually a list of limitations or informational items.**
> **EXAMPLE: The default values of maximum node and element name are 20000.**
>
> FUNCTION VERSION INFORMATION:
>
> **This tells you the version of the function to which the documentation applies. New versions (indicated by a higher number) may contain additional features requiring updated documentation. In this case, an updated version of the documentation for a specific function may be obtained from COSMO support.**
> **EXAMPLE: Any changes made to this function after version 1.00 01-93 will not be reflected in this release of the manual.**

1.4. Current Menu Structure

The following is a list of the entire COSMO menu structure showing Sub-Menu and Function numbers as well as the single-word commands (**NNNN**) to execute each function.

COSMO GEOMETRY (SUB-MENU 1)

1) COMBUSTOR LINER MODEL GENERATOR (**CLINER**)
   Reads a T/BEST Neutral FIle and generates a 3D Combustor Liner model.

2) COMPONENT SPECIFIC AIRFOIL GENERATOR (**AGEN**)
   Generates a user defined airfoil mesh from a geometric input file.

3) DISK MODEL GENERATOR (**DISK**)
   Generates a 2D and 3D disk model.

4) PRINT MENU STRUCTURE (**MENU**)
   Prints a copy of the current COSMO menu structure.

5) COSMO NEWS (**NEWS**)
   Prints news items about COSMO. Also prints phone list of COSMO personnel.

SIESTA FUNCTIONS (SUB-MENU 2)

1) CREATE SIESTA RDB FROM A UIF (**UIFREAD**)
   Reads a UIF and writes a SIESTA RDB.

2) RDB OUTLINE GENERATOR (**OUTLINE**)
   Determines the perimeter of discretized 2D regions and stores this information on a RDB. This information can be used by other functions.

3) RDB SURFACE GENERATOR (**SURFACE**)
   Determines the free surfaces, free edges and "boundary surfaces" of solid elements on a RDB. This information can be used to generate hidden-line and free-surface plots using the SIESTA Graphics function or to apply boundary conditions.

4) EDIT RANDOM DATA BASE (**EDIT**)
   Edits a RDB by adding data or overwriting existing data.

5) CREATE NASTRAN BULK DATA DECK (**NASTRAN**)
   Generates an NASTRAN bulk data deck from a RDB.

6) SIESTA GRAPHICS (**GRAPHICS**)
   Generates plots from a RDB. Extensive label and plot options are available including hidden line and free surface or edge plots. This function will also produce deformed

shape plots and contour plots of data from a Unified Output File. Also generates hard copies of plots (Hardcopy devices are system dependent but include Postscript.)

7) SIESTA PLOTTING (**PLOT**)
Generates X-Y plots of data. Includes log plots, polar plots, multiple axes, and hardcopy capability.

8) CONVERT RDB TO UIF (**UIFWRITE**)
Writes a UIF from a RDB. All data may be "dumped" or selective values may be written. This function also permits renumbering of nodes and elements on the output UIF.

9) CONVERT PATRAN NEUTRAL FILE TO UIF (**UF4**)
Converts PATRAN Neutral file to a Unified Input File.

10) MASTER REGION MESH GENERATOR (MR.MESH) (**MESH**)
Creates a discretized 2D mesh from a master region definition. Either triangles or quadrilaterals may be created.

11) 2D TO 3D MODEL GENERATOR (**TO3D**)
Generates a UIF of 3D elements from a RDB containing 2D data. For example, a 2D quadrilateral model may be rotated or stacked to create a 3D 8-noded brick model.

12) BANDWIDTH REDUCTION (GIBBS-POOLE-STOCKMEYER) (**BAND**)
Performs bandwidth reduction on a finite element model using the Gibbs-Poole-Stockmeyer algorithm.

13) TRANSLATE RDB TO IGES GEOMETRY FILE (**GEOM**)
Writes out geometry data from a RDB in IGES format.

14) TRANSLATE RDB TO PATRAN NEUTRAL FILE (**PATRAN**)
Generates a PATRAN Neutral File from an RDB.

15) AIRFOIL MESH GENERATOR (**AIRFOIL**)
Creates a user defined mesh from a variety of airfoil geometry input files.

16) SIESTA CSTEM DECK GENERATOR (**CP4**)
Generates an input file for the CSTEM analysis program from an RDB.

CUSTOMIZE COSMO (SUB-MENU 31)

1) BUILD CUSTOM MENUS
Permits users to define their own customized menus for COSMO.

2) DISPLAY CUSTOM MENU TITLES
Displays titles of user defined custom menus.

3) WRITE CUSTOM MENU DEFINITION FILE
Writes file of custom menu data for use in a later COSMO session.

Sub-Menus 32-50 are reserved as user custom sub-menus. These sub-menus are created using Sub-Menu 31, Function 1 or can be created by the default custom sub-menu definition file (cosmo.m) located in your current directory or in your COSMO.D directory. The COSMO.D directory should be located under your main or home directory. Once a custom sub-menu is created, it can be used like any other COSMO sub-menu. A custom sub-menu can contain up to twenty functions. The sub-menu title can be up to thirty characters and the function descriptions can be up to fifty characters. A single "*" is used to indicate the end of a sub-menu. Specific system commands can be entered in the function description for the the system function using # as a prefix. Also comments can append the function description for system function using $. For the example custom menu, entering "34 1" at the sub-menu, function prompt would result in the list of files in the current directory being printed to the terminal. The function description is: # ls -x $ List Directory. ($ List Directory is the comment)

Custom sub-menu definition file format:

```
Sub-Menu 32 Title
Function 1
    Function 1 description
Function 2
    Function 2 description
      . . .
      . . .
Function n
    Function n description
*
Sub-Menu 33 Title
Function 1
    Function 1 description
Function 2
    Function 2 description
      . . .
      . . .
Function n
    Function n description
```

Example cosmo.m: (Description of Data)

PROCESS 1 (Sub-Menu 32 Title)
11   (Function 1 - CLINER)
_   (Function description - C/R for COSMO default description)
AGEN (Function 2 - Airfoil Generator)
WONDERFUL COSMO AIRFOIL PROGRAM (Description of AGEN)
***   (End of Sub-Menu 32, Start of Sub-Menu 33)**
USER SYSTEM COMMANDS (Sub-Menu 33 Title)
SYSTEM   (Function 1)
 # ls -x $ List Directory   (List Directory in UNIX)
SYSTEM   (Function 2)

date (Print current date and time in UNIX)

If the example cosmo.m file is read into COSMO, entering two separate commands ("32 0", and "33 0") results in the available functions for each custom sub-menu being printed to the terminal.

```
                 YOU ARE IN THE MAIN SUB-MENU (0)
        ENTER DESIRED FUNCTION BY NUMBER OR ?,G,Q,SYSTEM,CRUN
32 0
              PROCESS 1 SUB-MENU (32)

               0 RETURN TO MAIN MENU (0)
CLINER         1 COMBUSTOR LINER MODEL GENERATOR
AGEN           2 WONDERFUL COSMO AIRFOIL PROGRAM

              YOU ARE IN THE PROCESS 1 SUB-MENU (32)
        ENTER DESIRED FUNCTION BY NUMBER OR ?,G,Q,SYSTEM
33 0
              USER SYSTEM COMMANDS SUB-MENU (34)

               0 RETURN TO MAIN MENU (0)
               1 # ls -x
               2 # date

              YOU ARE IN THE USER SYSTEM COMMANDS SUB-MENU (34)
        ENTER DESIRED FUNCTION BY NUMBER OR ?,G,Q,SYSTEM,CRUN
```

This should give the user a good idea of how custom sub-menus work in COSMO.

If you choose to create the custom sub-menus interactively, use Sub-Menu 31, Function 1. When you run this function, you will be prompted for a custom sub-menu file or to enter a carriage return for manual input. The custom sub-menu file is the same format at the cosmo.m file.

```
ENTER THE COSMO CUSTOM SUB-MENU 32 FILE
OR ENTER A CARRIAGE RETURN FOR MANUAL COMMAND INPUT
```

If you enter carriage return for manual input, you will be prompted for the sub-menu title:

```
ENTER THE CUSTOM SUB-MENU 32 TITLE
(UP TO 30 CHARACTERS)
ENTER * TO START SUB-MENU 33
```

You should enter the desired sub-menu title. Then you will be prompted for the function command for the first function in the custom sub-menu:

```
ENTER THE COSMO FUNCTION COMMAND FOR
CUSTOM SUB-MENU 32 FUNCTION 1
(ENTER A CARRIAGE RETURN TO END INPUT)
ENTER * TO START SUB-MENU 33
```

You then enter the desired function keyword or sub-menu/function of the function. Then you will be prompted for the function description to appear in the custom sub-menu:

```
ENTER THE COSMO FUNCTION CUSTOM DESCRIPTION
(UP TO 50 CHARACTERS),
ENTER CARRIAGE RETURN TO USE COSMO DESCRIPTION
```

You should enter the desired function description for the custom sub-menu. If you enter carriage return, the default COSMO function description is used. If the function is "SYSTEM", then the desired system command can be entered using the # prefix. In this case, when you select this function, the system command specified in the description will be executed.

You will be prompted for function commands and descriptions until you enter "*" to start a new custom sub-menu or carriage return at the function command prompt to end custom sub-menu input. Users will probably find it easier to create a cosmo.m file external to COSMO. However, Sub-Menu 31 is provided to assist users in creating custom sub-menus.

If you wish to print out the list of custom sub-menus, use Sub-Menu 31, Function 2. For the example custom sub-menus, the list of custom sub-menus is:

```
AVAILABLE CUSTOM SUB-MENUS
 32  PROCESS 1
 33  USER SYSTEM COMMANDS
```

If you wish to write out the custom sub-menus created by Sub-Menu 31, Function 1, use Sub-Menu 31, Function 3. If you run this function, the current custom sub-menu definition data is written to file 58. The file 58 can be renamed to cosmo.m for future use.

1.5. How to Run COSMO

In order to access the COSMO system, enter "xcosmo".

You can set up COSMO UNIX scripts to run specific functions without manual input. The script file is a list of commands you would give COSMO during any COSMO session. There are two methods used for COSMO scripts. One uses input redirection to read a list of commands from a file. For example, to create an SIESTA RDB from a UIF, the input file is (file name is file1):

```
UIFR
uif
QUIT
```

To run COSMO and create the SIESTA RDB, you would enter "xcosmo p <file1". Note: the argument p indicated that you are running production COSMO. The other method includes the "xcosmo" command in the script. This script to create an SIESTA RDB is (file name is file2):

```
xcosmo p <<stop01
UIFR
uif
```

QUIT
stop01

To run this COSMO script, you would enter "file2". (Note: file permissions must be set to allow for file execution or enter "sh file2" to run this script). If you do not understand these script methods, look up scripts in a UNIX User's Manual or call your system administrator for help. Scripts can may your work much easier and more automated. We recommend that you use them wisely.

Initially, the system will print out an opening banner, any currently applicable messages, and a list of available sub-menus. To enter a sub-menu, enter the appropriate sub-menu number. Upon entering a sub-menu from the main menu, a list of available functions is printed. The desired function is then selected by number.

In order to save the experienced user time and effort, the sub-menu and function number may be entered at any point. For example, if you were in the COSMO Main menu and wished to go the COSMO Combustor Liner Generator (Sub-Menu 1, Function 1 or **CLINER**), you would enter "1,1" or "1 1". This would take you directly to that function, and upon completion you would be in Sub-Menu 1.

Several other inputs are possible at any point. A carriage return will move you back up a level in the menu structure. This will take you back to the main menu from a sub-menu. A "Q" means quit and will return you to system level directly from anywhere. A "?" will print out the available functions if you are in a sub-menu or the available sub-menus if you are in the main menu.

The command "SYSTEM" allows you to issue a system command from within COSMO. This permits you to check your catalog for a forgotten file name, or perform other system functions without exiting COSMO. After entering "SYSTEM", you are prompted for the operating system command. You will remain in SYSTEM mode until you enter "QUIT".

All COSMO functions can be run by entering single-word commands. These commands are included in the Complete menu list of Section 1.4. Only the first four characters of these commands are required.

When files are written by COSMO, files referred to by two digits such as "60" have the file name "f60.dat". If COSMO needs to create a file for output, it will not overwrite or delete an existing file without permission. If, for example, a file 45 is needed for output, and you have a file named f45.dat in your current directory, you will have the option of deleting the existing file, making it a permanent file, or stopping execution. Also when entering file names, you can enter .dir to list the current directory or .sys to run a system command. You will then be prompted to enter the file name again.

If you screw up while entering a file name (e.g. entering an incorrect name), COSMO will prompt you for the file again after determining that the specified file doesn't exist. If you can't remember the proper file name, enter "QUIT". This will terminate the current function and return you to the sub-menu level.

When you run COSMO geometric recipe functions, you are prompted for the part name. This part name is used to name output files in COSMO. A part name may be up to eight characters. If the part name is entered as partname, then the COSMO output files would be partname.ext. The file extension used is different for each file type. The COSMO file extensions are:

.par	COSMO Parameter File
.igs	IGES File
.guf	Geometry UIF
.2uf	2D Model UIF
.3uf	3D Model UIF
.tuf	Temperature and Pressure UIF
.suf	Shell UIF
.pnf	Patran Neutral File
.cst	CSTEM Input File

1.6. COSMO Function Summaries

The COSMO function summaries are on the following pages.

COSMO FUNCTION SUMMARY

FUNCTION:

Combustor Liner Model Generator

SUB-MENU LOCATION:

COSMO Geometry (Sub-Menu 1, Function 1 or **CLINER**)

PURPOSE:

This function generates a 3D combustor liner model using a set of parameters to define the combustor liner cross section. Certain parameters can be read from a T/BEST Neutral File. Combustor inlet and exit pressures and temperatures can be used to map pressures and temperatures onto the combustor liner model. The output is a Unified Input File (UIF). Optionally an IGES file, Patran Neutral File and/or a CSTEM input deck can be generated.

INPUT FILE(S):

T/BEST Neutral File (55)
Combustor Liner Parameter File (52)

OUTPUT FILE(S):

Combustor Liner Parameter File (partname.par)
2D IGES file (partname.igs)
2D Geometry UIF (partname.guf)
2D Combustor Liner UIF (partname.2uf)
3D Combustor Liner UIF (partname. 3uf)
3D Pressure and Temperature UIF (partname.tuf)
3D Patran Neutral File (partname.pnf)
3D CSTEM input file (partname.cst)

REQUIRED USER INPUT:

The part name used to name output files must be input. The combustor liner parameters, number of fuel nozzles, number of circumferential elements and element circumferential spacing must be specified.

COMMENTS:

NONE

FUNCTION VERSION INFORMATION:

Any changes made to this function after version 1.02 06-94 will not be reflected in this release of the manual.

1.6.1.1.1. INTRODUCTION

The combustor liner model generator generates a finite element mesh of a combustor liner using a set of specified parameters to define the combustor liner cross section. Certain parameters can be read from a T/BEST Neutral File. Combustor inlet and exit pressures and temperatures can be used to map pressures and temperatures onto the combustor liner model. The output is a Unified Input File (UIF). Optionally a 2D geometry UIF, a 2D IGES file, a Patran Neutral File and/or a CSTEM input deck can be generated. The mesh will consist of 20-nodel solid elements.

When you run this function, you are prompted for the part name. This part name is used to name output files in COSMO. A part name may be up to eight characters. If the part name is entered as <u>partname</u>, then the 3D UIF would be <u>partname.3uf</u>.

1.6.1.1.2. COMBUSTOR LINER PARAMETERS

The combustor liner cross section is defined by 23 parameters. Figure 1.6.1.1.1 shows the combustor liner cross section and parameters. The parameters can be input interactively or using an input file. Also certain parameters can be set by the T/BEST Neutral File (see section 1.6.1.1.3).

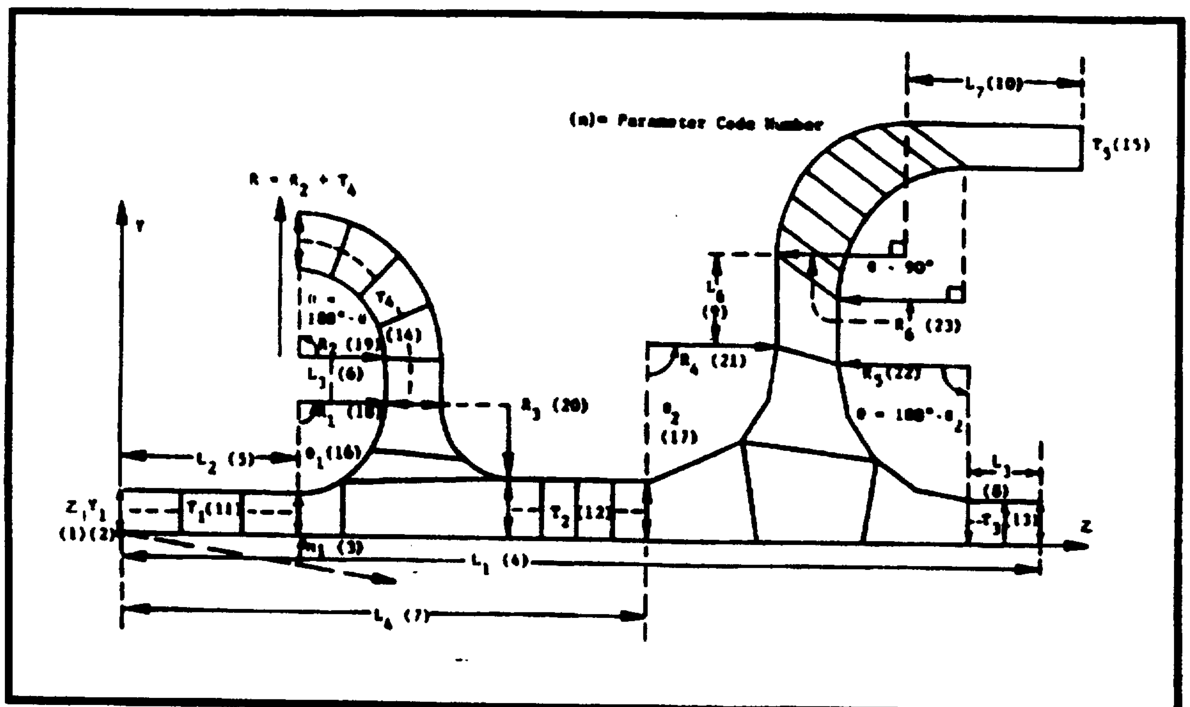


Figure 1.6.1.1.1
Combustor Liner Parameters

Z = Coordinate, Y = Coordinate, α = Rotation about Z, L = Length, T = Thickness, R = Radius of Curvature, θ = Angle of Rotation.

Combustor Liner Parameter List

Code	Name	Default	Code	Name	Default	Code	Name	Default
1	Z1	0.	2	Y1	10.	3	α1	0.
4	L1	10.5	5	L2	2.	6	L3	0.5
7	L4	6.	8	L5	0.8	9	L6	1.0
10	L7	2.	11	T1	0.5	12	T2	0.7
13	T3	0.5	14	T4	0.65	15	T5	0.5
16	θ1	90.	17	θ2	90.	18	R1	1.0
19	R2	1.	20	R3	0.75	21	R4	1.5
22	R5	1.5	23	R6	1.5			

There are also three control parameters:

24	IGES Write
25	Patran Neutral File Write
26	CSTEM Deck Write

The possible values for the control parameters are: -1 prompt user for the process, 0 skip the process, or 1 perform the process. The default values of these parameters is -1.

1.6.1.1.3. T/BEST NEUTRAL FILE

The T/BEST Neutral File is being used to define the engine component geometries and operating conditions using parameters. Data for the combustor is given for the component type: PBUR. Currently four geometric parameters and the inlet and exit pressures and temperatures can be read from the T/BEST Neutral File. The combustor liner inner radius (ROUT) is parameter 2, the combustor liner length (LENGTH) is parameter 4, and the combustor liner thickness (CTHK) is parameters 11, 12, 13, 14, and 15. Parameter 7 is calculated relative to parameter 4. The number of fuel nozzles is used to determine the model sector angle. This is discussed in section 1.6.1.1.4. Excerpts from the example T/BEST Neutral File is given in Table 1.6.1.1.1. The data that is read by this program is underlined. Note: this program can be updated to read additional data added to the T/BEST Neutral File.

```
*** TBEST EXECUTIVE SYSTEM - NEUTRAL FILE UPDATE ***
---------------------------------------------------
  .    .    .
  .    .    .
  .    .    .
ENGINE COMPONENT TYPE: PBUR   NCC          7
   MATERIAL                   CMPMAT     NICKEL
   PROCESS                    TYPROC     MAURER
   PBUR WEIGHT                WGHT        0.62710E+03   (lbs)
   STOCK MATERIAL WEIGHT      SWGHT       0.00000E+00   (lbs)
   MAURER WEIGHT FACTOR       MAURER      0.00000E+00   (lbs)
   COST TO MANUFACTURE ONE    COST1       0.00000E+00   ($)
   INNER RADIUS               RIN         0.19660E+02   (in.)
   OUTER RADIUS               ROUT        0.22750E+02   (in.)
   COMPONENT LENGTH           LENGTH      0.18000E+02   (in.)
   NUMBER OF NOZZLES          NCNOZZ         4.0
   COMBUSTOR THICKNESS        CTHK        0.10000E+00   (in.)
   .    .    .
   .    .    .
   .    .    .
```

```
GLOBAL VARIABLES 3 - MISSION
 ALTITUDE                      CALT       0.00000E+00    (ft.)
   SPEED                       V          0.00000E+00    (MACH No.)
 .    .    .
 .    .    .
 .    .    .
   COMPONENT TYPE: PBUR        NCC           7.0
   STATION NUMBER AT INLET     STIDIN        7.0
   PRESSURE AT INLET           PPBUR1     0.30870E+03    (psi.)
   TEMPERATURE AT INLET        TPBUR1     0.88948E+03    (F.)
   STATION NUMBER AT EXIT      STIDEX        8.0
   PRESSURE AT EXIT            PPBUR2     0.29018E+03    (psi.)
   TEMPERATURE AT EXIT         TPBUR2     0.26368E+04    (F)
 .    .    .
 .    .    .
 .    .    .
 ALTITUDE                      CALT       0.00000E+00    (ft.)
   SPEED                       V          0.00000E+00    (MACH No.)
 .    .    .
 .    .    .
 .    .    .
   COMPONENT TYPE: PBUR        NCC           7.0
   STATION NUMBER AT INLET     STIDIN        7.0
   PRESSURE AT INLET           PPBUR1     0.30352E+03    (psi.)
   TEMPERATURE AT INLET        TPBUR1     0.88194E+03    (F.)
   STATION NUMBER AT EXIT      STIDEX        8.0
   PRESSURE AT EXIT            PPBUR2     0.28531E+03    (psi.)
   TEMPERATURE AT EXIT         TPBUR2     0.25819E+04    (F)
 .    .    .
 .    .    .
 .    .    .
```

Table 1.6.1.1.1
T/BEST Neutral File

1.6.1.1.4. COMBUSTOR LINER MODEL GENERATION

Once the combustor liner parameters have been specified, the 2D cross section model is generated. You are prompted to enter 1 to write out the 2D IGES file. If you select this option, you will be asked for the part name, your name, and your organization. This data is written to the IGES file. The IGES file is written as partnam.igs. A separate 2D geometry UIF is written as partname.guf. Then the 2D model is rotated into a 3D sector model of the combustor liner. The 3D model is a symmetry model of half of one nozzle. The model is rotated about the Z axis. If the T/BEST Neutral File is not read, then you will be prompted for the number of nozzles and the number of elements circumferentially in the model. Then you will be prompted for the circumferential spacing. Enter a carriage return to use equal spacing. Enter circumferential percentages with the sum being less than 1.0. For example, for 4 circumferential elements, biasing percentages of 0.10 0.23 0.34 with place the nodal sections at 0, 10%, 33%, 67%, and 100% of the sector angle. The 3D combustor liner model is then generated.

If there is mission data in the T/BEST Neutral File, combustor inlet and exit pressures and temperatures are read from the file. A table of the mission data is written to the screen. The pressures are applied linearly along the inner surface of the combustor liner. There is no circumferential variation of pressure. The temperatures are also applied linearly to the nodes of the entire combustor liner. A circumferential temperature variation (varying with cos(θ)) can be superimposed on the linear temperature distribution. You will be prompted for the magnitude of the circumferential variation (DELT). This temperature variation is added to the linear temperatures.

1.6.1.1.5. OUTPUT

The output is a UIF (partname.3uf) of the combustor liner model. The model is a sector model with symmetry boundary conditions. Optionally a Patran Neutral File (partname.pnf) and/or a CSTEM input deck (partname.cst) can be written. If pressures and temperatures are read from the T/BEST Neutral File, the combustor liner pressures and pressures are written to a UIF (partname.tuf). You will be prompted for the case number of the mission data to use for the Patran Neutral File and/or the CSTEM input file. This allows pressures and temperatures to be written to these files.

1.6.1.1.6. EXAMPLE

Following is an example of running the Combustor Liner Model Generator. Note: this example uses the T/BEST Neutral File as input. See Figures 1.6.1.1.2 - 1.6.1.1.5 for plots of the combustor liner model generated.

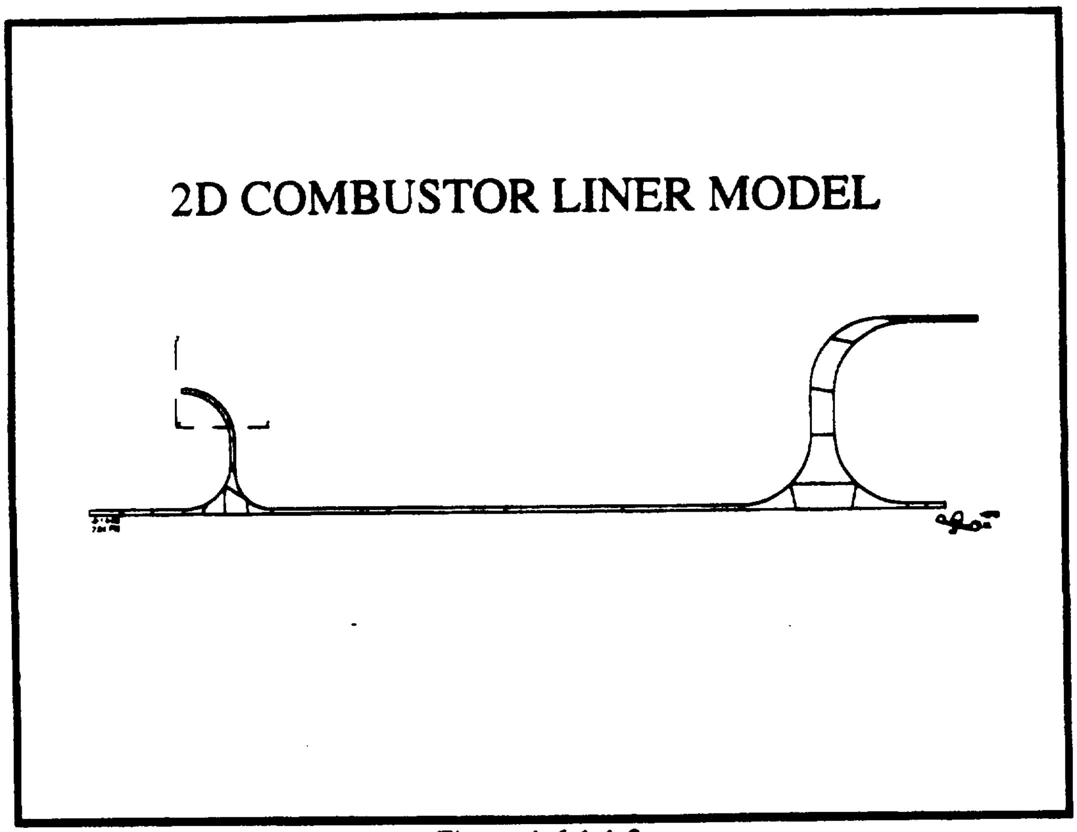


Figure 1.6.1.1.2

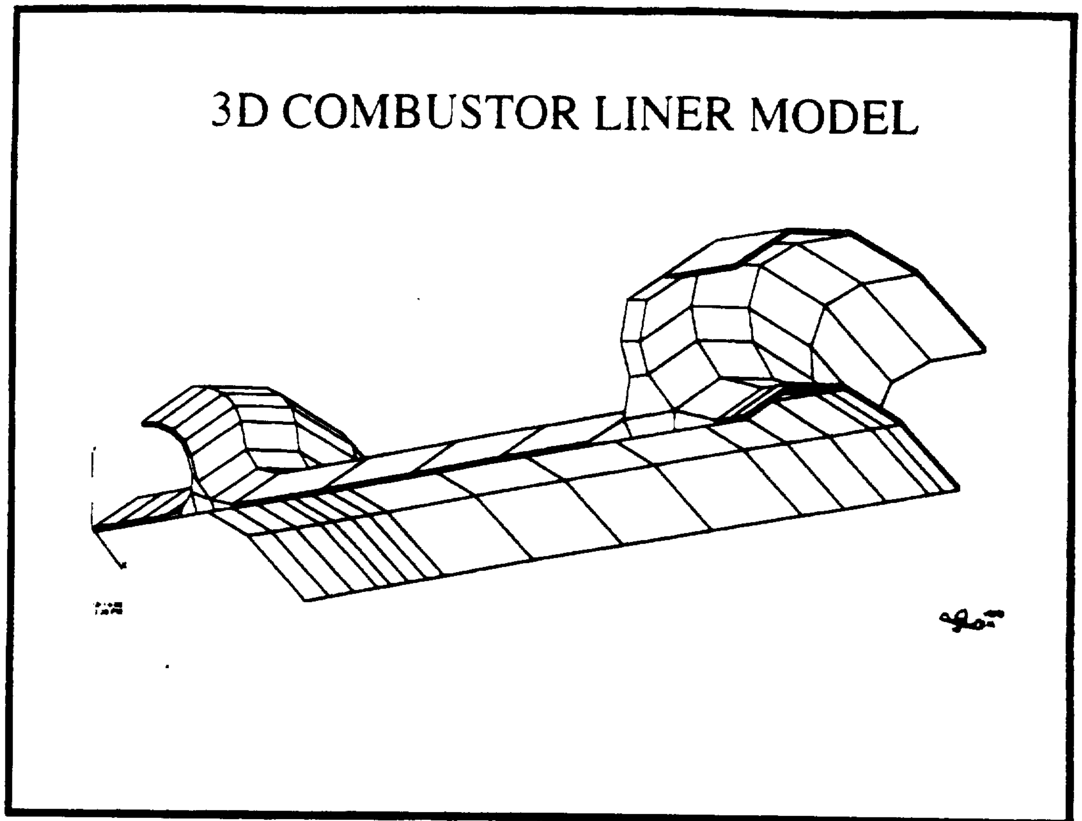


Figure 1.6.1.1.3

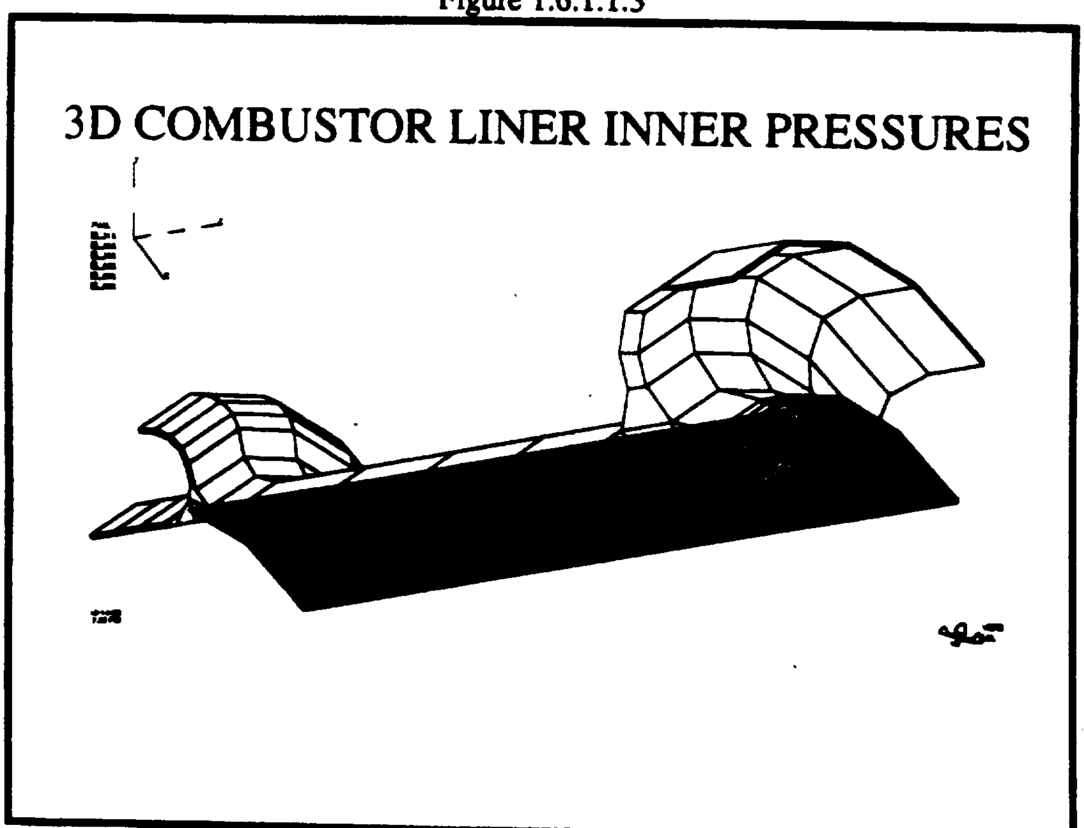


Figure 1.6.1.1.4

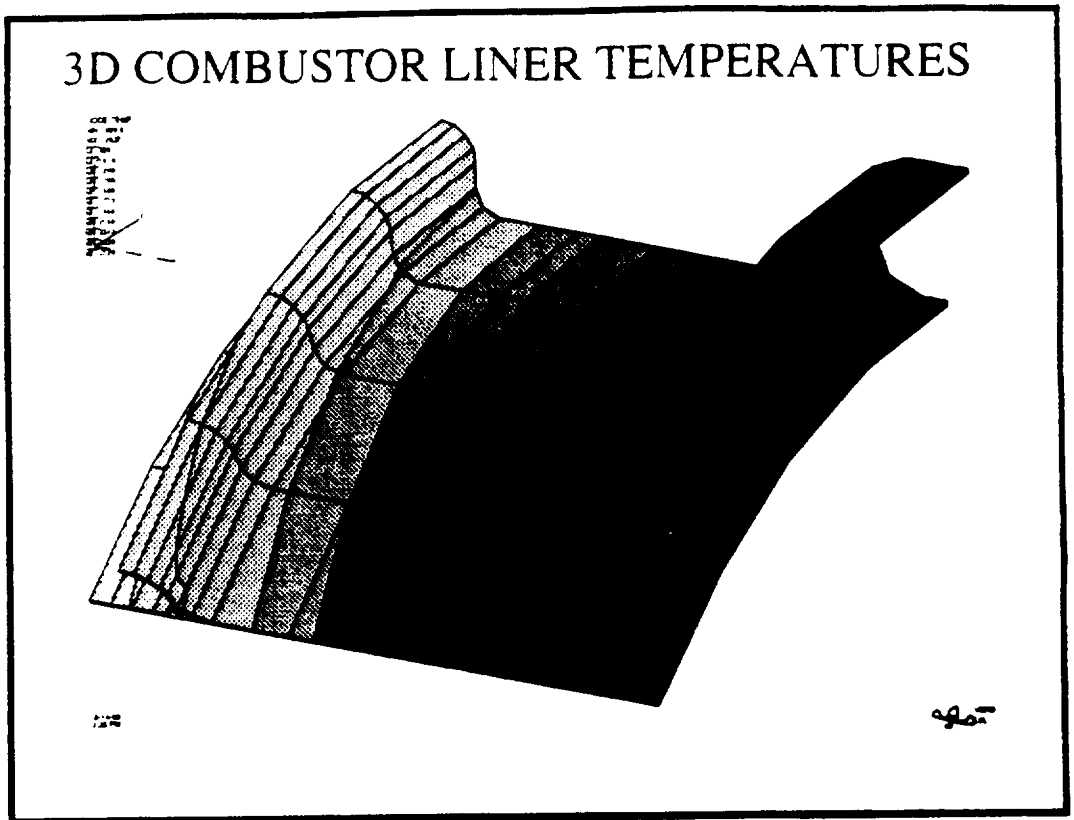


Figure 1.6.1.1.5

Combustor Liner Generator Example (Input is Underlined)

```
> xcosmo
    ##################################################################
                 CCCCC    OOOOOO    SSSSSS    MM    MM    OOOOOO
                 CC       OO  OO    SS        MMM MMM     OO  OO
                 CC       OO  OO    SSSSSS    MM M MM     OO  OO
                 CC       OO  OO        SS    MM M MM     OO  OO
                 CCCCCC   OOOOOO    SSSSSS    MM    MM    OOOOOO
    IT IS  4:32 PM ON 06-30-94 SYSTEM c0401         VERS I.01 06-30-94
    ##################################################################
     This is Version I.01 of COSMO (Production).
     Type NEWS for the latest COSMO News - last update 06-29-94.
          Type NEWS for more information.

                   AVAILABLE SUB-MENUS
    (?)  MENU, (Q)  QUIT, (SYSTEM) SYSTEM
     0 EXIT FROM COSMO

     1 COSMO GEOMETRY                  2 SIESTA FUNCTIONS
    31 CUSTOMIZE COSMO

         CURRENT MAXIMUM SUB-MENU IS 31

         YOU ARE IN THE MAIN MENU (0)
    ENTER DESIRED SUB-MENU BY NUMBER OR ?,G,Q,SYSTEM
CLIN
    ############################################################
        COSMO COMBUSTOR LINER GENERATOR  VERS I.02  06-30-94
              ON c0401          AT  4:32 PM 06-30-94
    ############################################################

    ENTER THE PART NAME (UP TO 8 CHARACTERS)
```

```
part1

     ENTER THE T/BEST NEUTRAL FILE NAME
     OR ENTER "NONE" TO SKIP READING NEUTRAL FILE
neufile

     COMBUSTOR LINER DATA SET BY T/BEST NEUTRAL FILE:
     COMBUSTOR LINER INNER RADIUS =  22.750 IN
     COMBUSTOR LINER LENGTH       =  18.000 IN
     COMBUSTOR LINER THICKNESS    =    .100 IN
     NUMBER OF FUEL NOZZLES       =   4

     THE CURRENT COMBUSTOR LINER PARAMETERS ARE:
          CODE       VALUE                   CODE       VALUE
           1          .00000                  2        22.75000
           3          .00000                  4        18.00000
           5         2.00000                  6          .50000
           7        13.70000                  8          .80000
           9         1.00000                 10         2.00000
          11          .10000                 12          .10000
          13          .10000                 14          .10000
          15          .10000                 16        90.00000
          17        90.00000                 18         1.00000
          19         1.00000                 20          .75000
          21         1.50000                 22         1.50000
          23         1.50000                 24        -1.00000
          25        -1.00000                 26        -1.00000

     ENTER PARAMETER CHANGES (ENTRY CODE, NEW VALUE)
     OR <CR> TO GENERATE THE COMBUSTOR LINER PE2DS
     OR "FILE" TO ENTER PARAMETERS FROM A FILE
     OR "LIST" TO LIST OF THE PARAMETER VALUES
     OR "QUIT" TO QUIT
<CR>

     THE COMBUSTOR LINER PARAMETER FILE IS  part1.par
     ENTER 1 TO GENERATE AN IGES FILE
1
     ##############################################################
        UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
     ##############################################################
 STORAGE   VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
 READER    VERS I.04  06-02-94

     INPUT FILE HAS BEEN PROCESSED
     YOUR RANDOM DATA BASE IS ON FILE  37
     ##############################################################
          SIESTA GEOMETRY RECREATOR    VERS I.01  05-05-94
     ##############################################################
 DATA BASE VERS I.00  04-11-94
     GEOM DATA WILL BE CONVERTED
     REQUIRED INFORMATION FOR IGES FILE HEADER
     ENTER THE PART NAME (UP TO 10 CHAR.):
CLINER
     ENTER YOUR NAME (UP TO 20 CHAR.):
COSMO TEST
     ENTER YOUR ORGANIZATION NAME (UP TO 20 CHAR.):
GEAE
          22 GENTS WRITTEN
     THE IGES GEOMETRY DATA IS ON FILE  40
     THE MODEL GEOMETRY UIF IS part1.guf

     THE MODEL IGES FILE IS part1.igs


     THERE ARE    4 FUEL NOZZLES
```

```
    ENTER THE NUMBER OF CIRCUMFERENTIAL ELEMENTS TO USE
4
    ENTER THE   3 CIRCUMFERENTIAL BIASING PARAMETERS
    ENTER AS PERCENTS, THE SUM BEING LESS THAN 1.0
    ENTER CARRIAGE RETURN FOR EQUAL SPACING
<CR>

    ##############################################################
       UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
    ##############################################################
 STORAGE   VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
 READER    VERS I.04  06-02-94

    INPUT FILE HAS BEEN PROCESSED
    YOUR RANDOM DATA BASE IS ON FILE  37
    ##############################################################
       SIESTA 2D TO 3D MODEL GENERATOR  VERS I.01  06-23-94
    ##############################################################
 DATA BASE VERS I.00  04-11-94
    ENTER STRUCTURAL MODEL PROPAGATION OPTION
          (Q) - EXIT FROM 2D TO 3D, (?) - MENU, (INFO) - INFORMATION
          (0)  DO NOT PROPAGATE TEMPERATURES, MATERIAL CODES
          (1)  PROPAGATE TEMPERATURES FROM 2D TO 3D MODEL
          (2)  PROPAGATE MATERIAL CODES FROM 2D TO 3D MODEL
          (3)  PROPAGATE BOUNDARY SURFACES FROM 2D TO 3D MODEL
          (4)  SHIFT PARENT LAYER NODE AND ELEMENT NAMES
          (5)  DISTRIBUTE POINT WEIGHTS OVER THE ROTATED MODEL
 2. 3.
    ENTER GENERATION OPTION
          (Q) - EXIT FROM 2D TO 3D, (?) - MENU, (INFO) - INFORMATION
          (0) ROTATE A 2D MODEL ABOUT AN AXIS
          (1) STACK A 2D MODEL ALONG AN AXIS
          (2) FILL BETWEEN TWO SURFACES
     0
    YOUR MODEL HAS COORDINATES IN ALL THREE AXES.
    ENTER COORDINATE DESCRIPTION OPTION
          (Q) EXIT 2D TO 3D
          (0) THE MODEL IS IN THE YZ PLANE
          (1) THE MODEL IS IN THE XZ PLANE
          (2) THE MODEL IS IN THE XY PLANE
     0
    ENTER DESIRED ROTATION OPTION
          (Q) - EXIT FROM 2D TO 3D, (?) - MENU, (INFO) - INFORMATION
          (0) EQUALLY SPACED ROTATION
          (1) UNEQUALLY SPACED ROTATION
          (INFO) FOR MORE INFORMATION
     0
    ENTER DESIRED SYMMETRY BOUNDARY CONDITION OPTION
          (0) APPLY NO BOUNDARY CONDITIONS
          (1) APPLY SYMMETRIC BOUNDARY CONDITIONS
          (2) APPLY ANTI-SYMMETRIC BOUNDARY CONDITIONS
          (3) APPLY PSEUDO SYMMETRIC BOUNDARY CONDITIONS
     1
    YOU HAVE SELECTED EQUALLY SPACED ROTATION.
    ENTER THE AXIS YOU WISH TO ROTATE ABOUT AS Y OR Z
 Z
    ENTER THE NUMBER OF ELEMENT LAYERS, THE ANGLE BETWEEN LAYERS,
          THE ANGLE OF THE PARENT LAYER, THE ANGLE OF TWIST,
          THE NODE NAME ADDER, AND THE ELEMENT NAME ADDER
 4. 11.25
                    200 NODES PROCESSED
                    400 NODES PROCESSED
                    600 NODES PROCESSED
                    800 NODES PROCESSED
                    914 NODES PROCESSED
    GENERATING SYMMETRY BOUNDARY CONDITIONS
```

```
          0 NODAL ZERO DISPLACEMENTS PROPAGATED
              27 PE2DS PROCESSED INTO       108 VANSS
     YOUR NEW 3D UIF IS ON FILE 31
     ##############################################################
         UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
     ##############################################################
 STORAGE    VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
 READER     VERS I.04  06-02-94

     INPUT FILE HAS BEEN PROCESSED
     YOUR RANDOM DATA BASE IS ON FILE  37

     THE COMBUSTOR LINER 2D MODEL FILE IS part1.2uf
     THE COMBUSTOR LINER 3D MODEL FILE IS part1.3uf

 DATA BASE VERS I.0C  04-22-93

     THE COMBUSTOR LINER MODEL WEIGHT IS    32.19     LB
     THE TOTAL COMBUSTOR LINER WEIGHT IS    257.5     LB


     COMBUSTOR LINER MISSION DATA SET BY T/BEST NEUTRAL FILE:
  CASE   ALTITUDE    SPEED       TINLET      TEXIT       PINLET       PEXIT
    1  0.0000E+00  0.0000E+00    889.5       2637.       308.7        290.2
    2  0.0000E+00  0.0000E+00    881.9       2582.       303.5        285.3
    3  0.0000E+00   .2000        899.0       2652.       325.0        305.5
    4    689.0      .3000        905.2       2670.       328.9        309.2
    5   2000.       .4000        911.4       2684.       330.2        310.4
    6  1.0000E+04   .6000        887.3       2632.       279.4        262.6
    7  2.0000E+04   .9000        892.9       2647.       247.8        233.0
    8  3.0000E+04  1.050         850.8       2556.       190.3        178.8
    9  3.6089E+04  1.400         948.5       2776.       218.3        205.2
   10  4.0000E+04  1.600         1057.       3021.       230.6        216.7
   11  4.0000E+04  1.630         1069.       3030.       249.7        234.8
   12  5.0000E+04  1.800         1117.       3027.       181.4        170.6
   13  5.2500E+04  2.000         1196.       3034.       197.7        185.8
   14  5.5000E+04  2.200         1248.       3035.       201.1        189.0
   15  6.0000E+04  2.400         1250.       3035.       163.5        153.7

    ENTER THE CIRCUFERENTIAL TEMPERATURE VARIATION MAGNITUDE
100
    READING A CASE OF TEMPERATURES AND PRESSURES
    THE CASE NUMBER MUST BE SPECIFIED
     ##############################################################
        SIESTA RANDOM DATA BASE EDITOR   VERS I.01  05-25-94
    ##############################################################
 STORAGE    VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
 READER     VERS I.04  06-02-94
    CASE       1 ENCOUNTERED
    ENTER DESIRED CASE NUMBER TO BE READ
    OR ENTER 'QUIT' TO EXIT READING THE UIF
1
    END OF CASE       1
    ENTER 'Q' TO EXIT READING THE UIF
          'F' TO RUN A SIESTA FUNCTION
          'R' TO READ ADDITIONAL CASES
q

    INPUT FILE HAS BEEN PROCESSED
    EDITING OF RANDOM DATA BASE IS COMPLETE

    THE COMBUSTOR LINER MISSION TEMPERATURES AND PRESSURES
    FOR  15 CASES ARE IN  part1.tuf

    ENTER 1 TO GENERATE A PATRAN NEUTRAL FILE
```

```
1
    ################################################################
        SIESTA PATRAN INPUT GENERATOR     VERS I.00  04-11-94
    ################################################################
 DATA BASE VERS I.00  04-11-94
    ENTER THE PATRAN NEUTRAL FILE TITLE (MAX 40 CHARACTERS)
title
    ENTER THE ANALYSIS CODE:
    1-ANSYS  2-NASTRAN  3-SIESTA  4-PTHERMAL  5-UNIGRAPHICS
1
 $
 $   TOTAL DATA BASE CONTENTS
 $            FIRST     LAST      MIN      MAX   NUMBER
 $    NODES       1     2538        1     2538      914
 $    ELEMS       0        0        1      108      108
 $    BTABS       0        0        1       20       20
 $    VANSS       1      108        1      108      108
 $
    WRITING PATRAN NEUTRAL FILE
         200 NODES WRITTEN
         400 NODES WRITTEN
         600 NODES WRITTEN
         800 NODES WRITTEN
         914 TOTAL NODES WRITTEN
         200 NODAL TEMPERATURES WRITTEN
         400 NODAL TEMPERATURES WRITTEN
         600 NODAL TEMPERATURES WRITTEN
         800 NODAL TEMPERATURES WRITTEN
         914 TOTAL NODAL TEMPERATURES WRITTEN
         276 SETS OF NODAL DISPLACEMENTS WRITTEN
         108 TOTAL VANS WRITTEN
          64 VANS FACE NORMAL PRESSURES WRITTEN

    YOUR PATRAN NEUTRAL FILE IS ON FILE  35
    THE MODEL PATRAN NEUTRAL FILE IS part1.pnf

    ENTER 1 TO GENERATE A CSTEM INPUT FILE
1
    ################################################################
          SIESTA CSTEM DECK GENERATOR  VERS I.00  04-11-94
    ################################################################
 DATA BASE VERS I.00  04-11-94
         TOTAL DATA BASE CONTENTS
        DATA TYPE          MIN     MAX    TOTAL
    ------------------   ------  ------  ------
          NODE               1    2538     914
          VANS               1     108     108

    ENTER THE ANALYSIS IDENTIFICATION (UP TO 80 CHAR.)
Title
         200 NODES WRITTEN
         400 NODES WRITTEN
         600 NODES WRITTEN
         800 NODES WRITTEN
         914 NODES WRITTEN
      108 20-NODED BRICKS WRITTEN
    ENTER THE LAYER SPECIFICATION DATA FILE NAME
    THIS DATA WILL BE ADDED TO THE CSTEM DECK
    OR ENTER 'NONE' TO SKIP ELEMENT LAYERING
NONE
         276 NODAL ANGLE LINES WERE WRITTEN
         276 NODAL FIXITY LINES WERE WRITTEN
      914 NODAL TEMPERATURES WRITTEN
         224 ELEMENT PRESSURES WRITTEN
    YOUR CSTEM DECK IS ON FILE 35
    THE MODEL CSTEM INPUT FILE IS part1.cst
```

```
     YOU ARE IN THE MAIN MENU (0)
ENTER DESIRED SUB-MENU BY NUMBER OR ?,G,Q,SYSTEM
E

##########################################################################
  IT IS NOW  4:33 PM ON 06-30-94 WE THANK YOU FOR YOUR PATRONAGE.
##########################################################################
```

COSMO FUNCTION SUMMARY

FUNCTION:

Component Specific Airfoil Generator

SUB-MENU LOCATION:

COSMO Geometry (Sub-Menu 1, Function 2 or **AGEN**)

PURPOSE:

This function generates 20-noded solid element meshes from an aero coordinate definition. The output is a Unified Input File (UIF). Optionally a Patran Neutral File and/or a CSTEM input deck can be generated.

INPUT FILE(S):

Aero Geometry file (40)

OUTPUT FILE(S):

UIF (afuif)
Patran Neutral File (afpnf)
CSTEM input file (cstdeck)

REQUIRED USER INPUT:

The part name used to name output files must be input. The chordwise, spanwise, and thickness mesh densities must also be specified.

COMMENTS:

This function is particularly useful for the generation of 3D airfoils. (Fan and compressor vanes and blades).

FUNCTION VERSION INFORMATION:

Any changes made to this function after version L02 06-94 will not be reflected in this release of the manual.

1.6.1.2.1. INTRODUCTION

The airfoil mesh generator generates a finite element mesh of user specified size from either aero coordinate data in the form of 'glass master' sections. The number of sections used is arbitrary and does not have to correspond to the desired airfoil mesh size. You have control over mesh density, spacing, and input modification. The mesh will consist of 20-nodel solid elements.

When you run this function, you are prompted for the part name. This part name is used to name output files in COSMO. A part name may be up to eight characters. If the part name is entered as partname, then the 3D UIF would be partname.3uf.

The program refers to the outer part of an airfoil as the tip, and the inner part of an airfoil as the root. The sides of the airfoil are called pressure and suction, the pressure side is on the right when the airfoil is viewed from forward looking aft. For shrouds the tip is in the normal direction of rotation.

This program works in a coordinate system that is relative to the airfoil. The 'CHORDAL' direction is a CARTESIAN axis that is positive when traveling from the leading edge to the trailing edge (this is not the airfoil chord). The 'SPAN' direction is a CARTESIAN axis that is positive when traveling from the root to the tip. The 'THICKNESS' direction is a CARTESIAN axis that is positive in a right hand system with 'SPAN' crossed into 'CHORDAL'. See Figure 1.6.1.2.1.

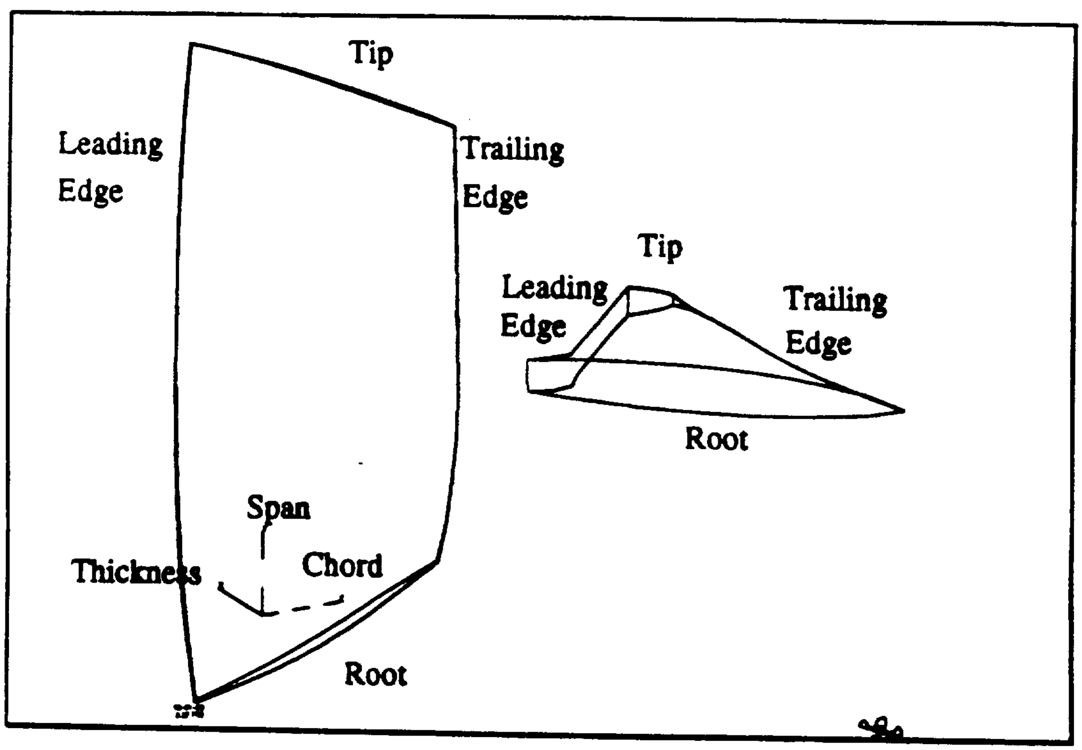


Figure 1.6.1.2.1
Coordinate system

1.6.1.2.2. AIRFOIL FILE DEFINITION

The DIGITIZED GLASS MASTER FILE is the input file format used by this program. This format will allow almost anything to be modeled as an airfoil. This form has a constant even number of pairs per section. The ordering is pairs of pressure and suction side points starting at the leading edge (see Figure 4.17.3.2). Each point has three coordinates, the first is 'SPAN', the second is 'THICKNESS', and the third is minus 'CHORDAL'. See Table 1.6.1.2.1. You must tell the program the number of pairs per section and the number of sections.

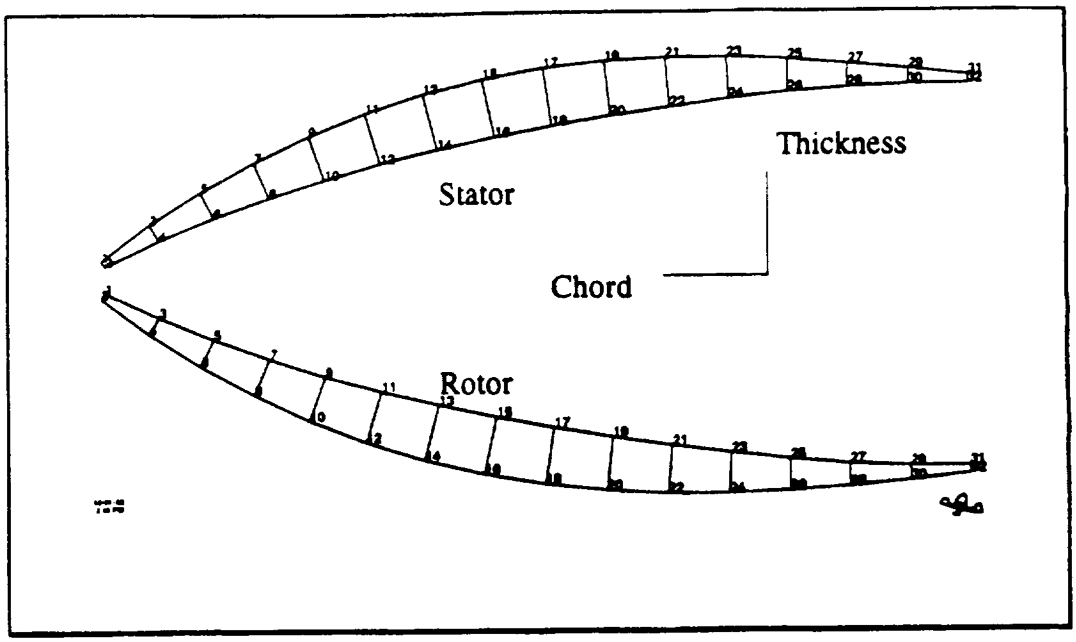


Figure 1.6.1.2.2
Digitized node order

26.0035	-.08969	-1.41029
26.0035	-.16855	-1.405
26.0035	-.09033	-1.47949
26.0035	-.15928	-1.47576
26.0035	-.08998	-1.54177
26.0035	-.15058	-1.5394
26.0035	-.08894	-1.59711
26.0035	-.14269	-1.59594
26.0035	-.08728	-1.64899
26.0035	-.13529	-1.64895
26.0035	-.08006029	-1.8013
26.0035	-.11184	-1.80471
26.2688	.548334	1.518
26.2688	.526686	1.53315
26.2688	.48002	1.37991
26.2688	.4272	1.4106
26.2688	.45869	1.33374
26.2688	.39743	1.36754
26.2688	.43651	1.28423
26.2688	.36618	1.32126

Table 1.6.1.2.1
Digitized Glass Master File

1.6.1.2.3. MODIFYING THE INPUT

There are four input modifications in the mesh generator:

1) Scaling. Glass masters usually are enlarged views of airfoil sections. All of the input coordinates in the aero file are divided by the factor which you enter here. The default scale is 1.

2) Engine and Airfoil Offsets. Digitized data is often entered relative to some airfoil local coordinate system (usually the origin is on the airfoil). This modification allows you to offset from the airfoil local system to the Tilt and Lean coordinate system (usually the origin is on the dovetail), tilt and lean the airfoil, and offset again to the Engine coordinate system (usually the origin is on the engine centerline). See Figure 1.6.1.2.3.

   A total of eight values can be entered here.

   The first three are THICKNESS, SPAN, CHORD offsets added to the rotated airfoil to convert from the Tilt and Lean system to the Engine system (these are the offsets to use if there is no tilt or lean).

   The next three values are THICKNESS, SPAN, CHORD offsets subtracted from the Airfoil coordinates to convert to the Tilt and Lean system (necessary if the tilt and lean are done about 'arbitrary' axes).

   The last two values are the Tilt (about the CHORD), and the Lean (about the THICKNESS). The Airfoil is Tilted then Leaned.

3) Section Switch. Because of differences in terminology between compressor vs. turbine or stator vs. rotor or fixed stator vs. counterrotating aero groups, aero files often have the pressure and suction sides reversed. The result of this will be negative volume elements. A nondefault response will switch the pressure and suction sides.

   Tip Root Switch. All of the input file types have a default definition of whether the sections are entered from the tip or the root. If you have a file that does not match the default for the file type you are using (you get upside down models) you will need this. Enter a nondefault response to switch the order of the sections.

4) Leading Edge/Trailing Edge Averaging. Aero supplies extremely accurate airfoil coordinates. One problem this causes is that the program uses the first and last pairs points for the leading and trailing edges. This can cause distorted edge elements (See Figure 1.6.1.2.4). If this is a problem, you must enter the number of pairs of points for the leading and trailing edge that the program should use to calculate an effective area. The edge thicknesses are modified to maintain this area, ignoring the intermediate pairs of points.

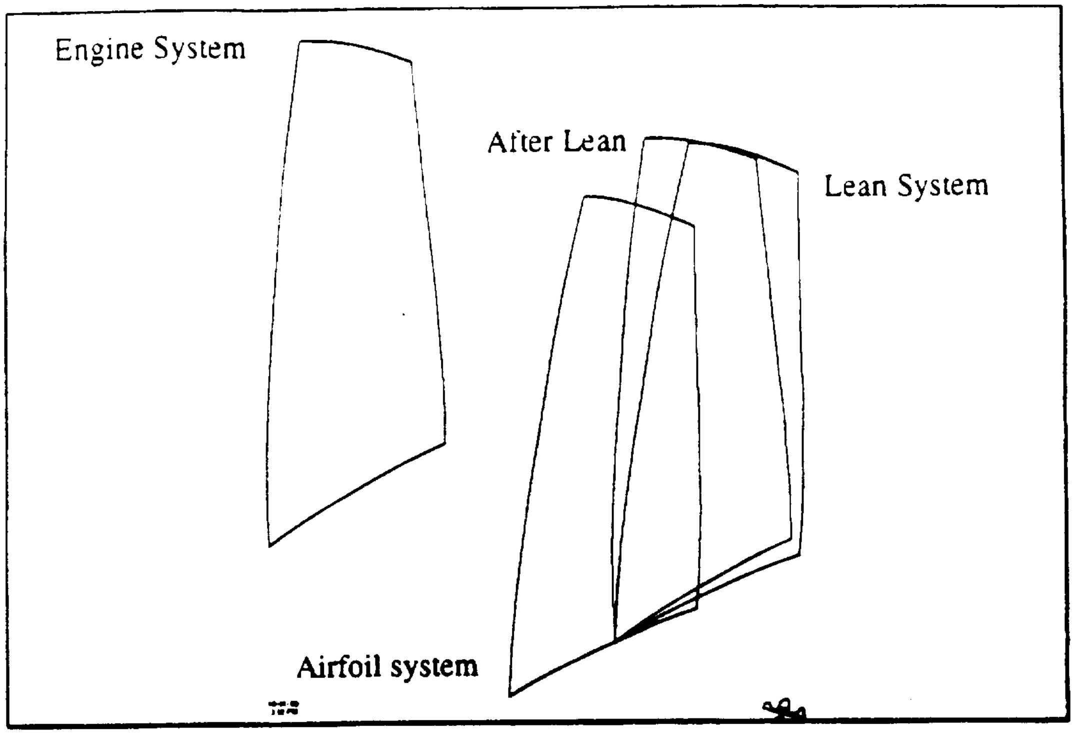


Figure 1.6.1.2.3
Tilt/Lean Coordinate Systems

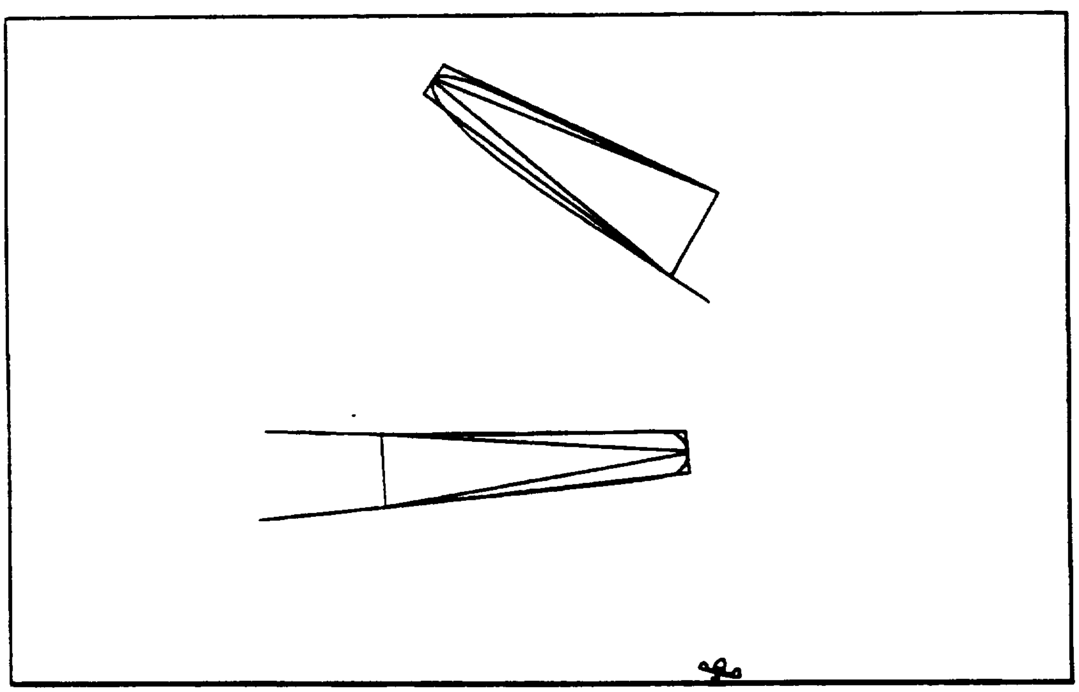


Figure 1.6.1.2.4
Leading Trailing Edge Correction

1.6.1.2.4. CHORD ELEMENT DENSITY AND SPACING

You enter the number of elements in the chord direction. The chord spacing is calculated using weightings. There are two options for generating the weighting values.

Even spacing (0) is the easiest.

User defined weighting (3) is for cases where there is some need for you to control the spacing. You should enter weightings between 0.0 (leading edge) and 1.0 (trailing edge) (note that 0. and 1. are implied) that define the spacing (see Figure 1.6.1.2.5). You may change weightings for each section or use the same spacing for all sections. If you need to enter more numbers than will fit on a single line, end the line with an ampersand (&) and continue on the next line.

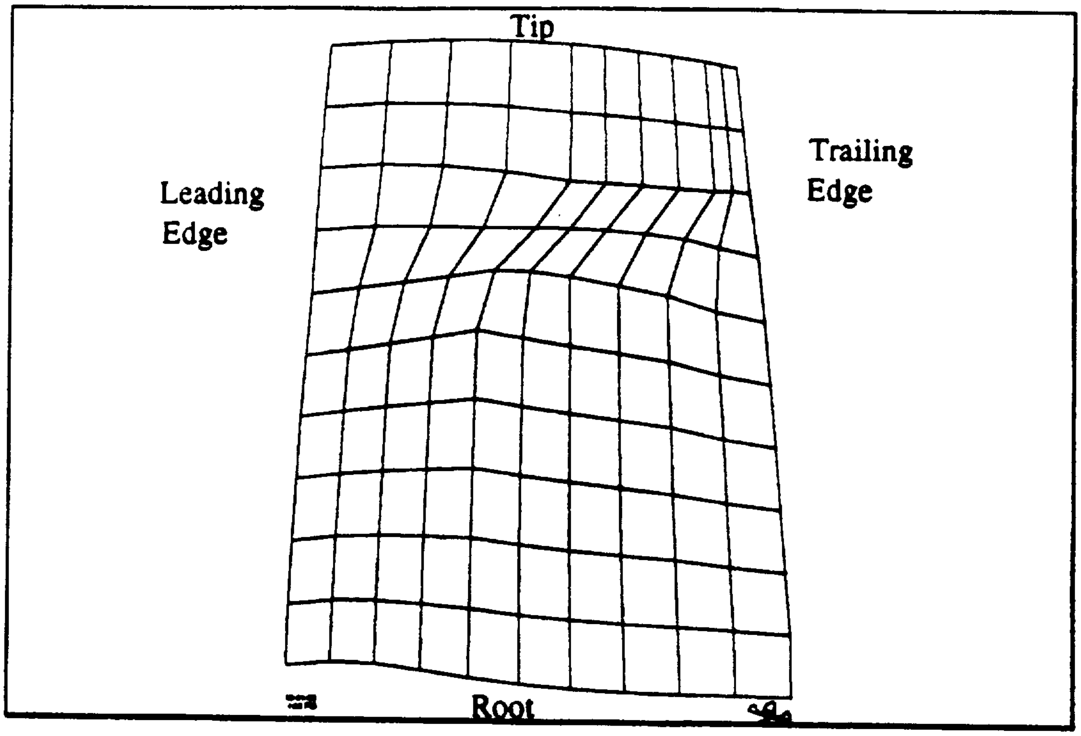


Figure 1.6.1.2.5
Example of User Weighting

1.6.1.2.5. SPAN ELEMENT DENSITY

You must specify the number of elements to be generated in the span direction.

1.6.1.2.6. MULTIPLE ELEMENTS THRU THE THICKNESS

The number of elements thru the thickness must be specified. If one (the default) or two elements thru the thickness are specified no other information is required, even spacing is used. For three of more elements thru the thickness you have three options as to how the thickness spacing is varied. The first options is equally spaced layers, the airfoil is divided evenly (see Figure 1.6.1.2.6).

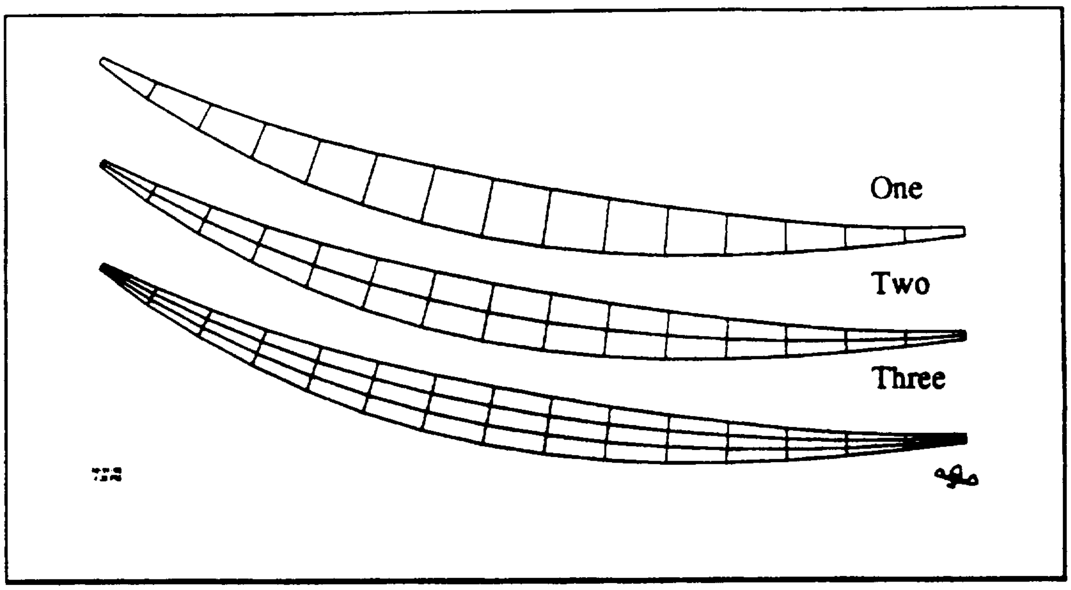


Figure 1.6.1.2.6
Even Spacing of Elements Thru the Thickness

The second option is adding coatings to the airfoil. N-2 coatings may be added to the pressure side (see figure 1.6.1.2.7). N-1-the number of pressure side coatings may be added to the suction side. This leaves at least one layer in the pressure side, the suction side and the 'Real' airfoil. Aside from ridiculous looking airfoils, no errors can be generated with this option.

The third option is subtracting coatings off of the airfoil surface (see Figure 1.6.1.2.7). N-2 coatings may be subtracted from the pressure side. N-1-the number of pressure side coatings may be subtracted from the suction side. This leaves at least one layer in the pressure side, the suction side and the 'real' airfoil. (Called 'real' from the lamination case where this is the starting form).

For the subtraction option if the specified thicknesses for the pressure and suction sides are greater than the aero airfoil thickness the 'real' airfoil thickness would have to be negative. For this negative 'real' thickness problem the user is prompted for a minimum 'real' thickness (it should be obvious that positive but tiny real thickness are as bad as negative thicknesses.)

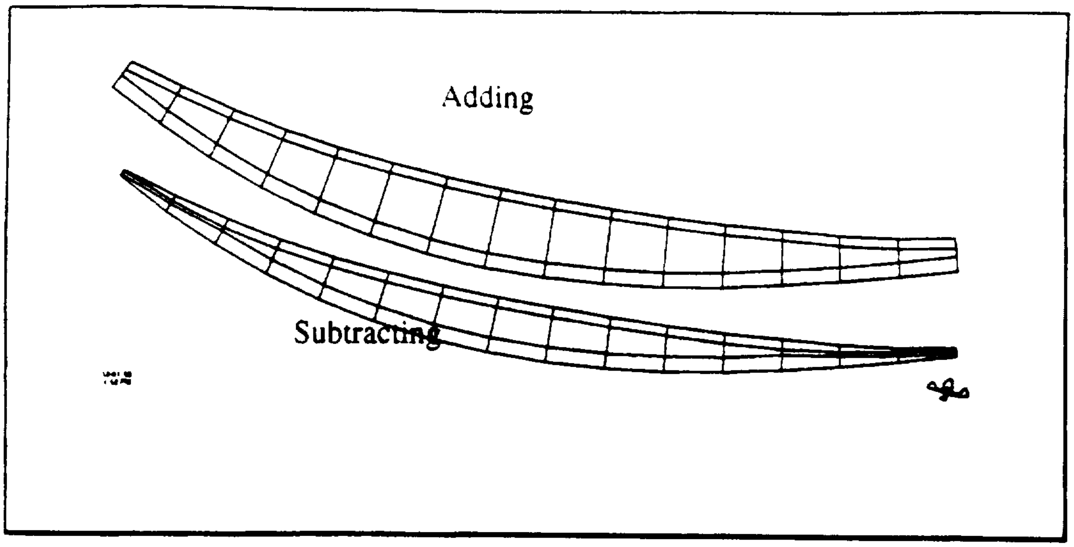


Figure 1.6.1.2.7
Adding and Subtracting Airfoil Coatings

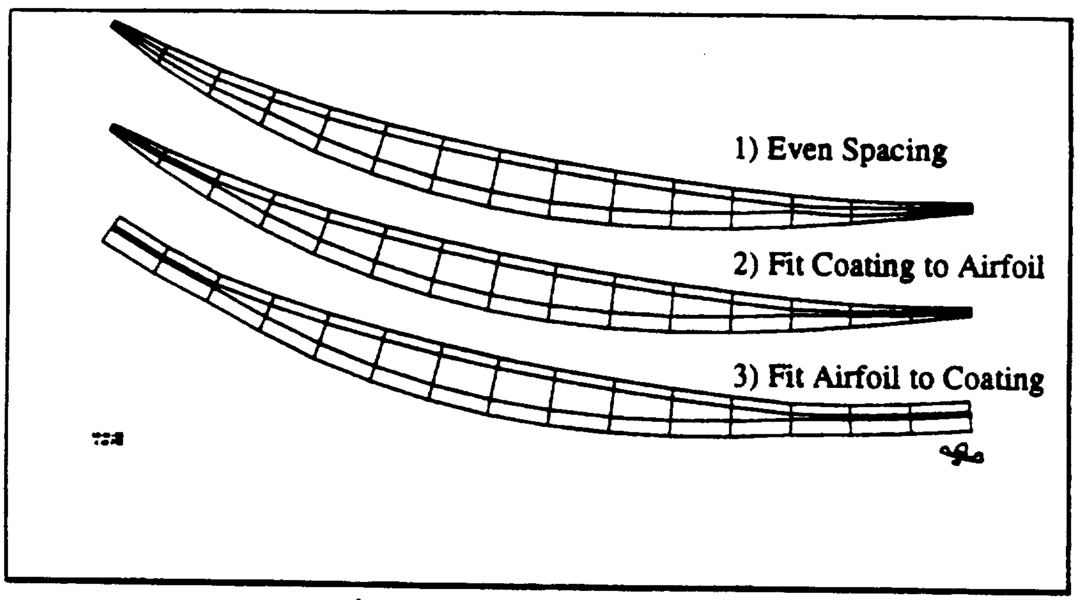


Figure 1.6.2.1.8
Coating Thickness Corrections

For each point on the airfoil surface the thickness is checked and several adjustments can be made if the airfoil is too thin for the thickness requested (see Figure 1.6.2.1.8).

0 No correction, if you are sure the airfoil will always be thicker than the coatings.

1 Even spacing, for this point generate N evenly spaced layers (this may causes coatings to vary wildly from one point to the next).

2 Ratio the coatings to match the airfoil, the 'real' thickness will total to match the minimum real thickness, the overall thickness will total to match the aero thickness. The pressure and suction side thicknesses for this point will not be what was entered.

3 Change the airfoil to match the coatings, the 'real' thickness will total to match the minimum real thickness, the overall thickness will total to the sum of the pressure and suction side thickness plus the minimum real thickness. The airfoil thickness will not be the aero thickness.

4 Enter new coatings, new coatings can be entered for this point.

Correction options 0, 1, 2 and 4 maintain the outer airfoil contour. Options 2 and 3 maintain the 'real' airfoil contour.

Once the number of elements thru the thickness has been established, you will be prompted for the thickness rows and chord rows of elements to eliminate to simulate a hollow airfoil. For example, assume there are 4 elements thru the thickness and 10 elements in the chord direction. Specifying thickness rows 2 and 3 and chord locations 4 and 7, results in the middle two rows of elements being missing for chord locations 4 and 7.

1.6.1.2.7. OUTPUT

The output is a UIF (partname.3uf) of the airfoil model. For airfoils the 'CHORDAL' direction will be axial (Z), the 'SPAN' direction will be radial (Y), and the 'THICKNESS' direction will be tangential (X). For shrouds the 'CHORDAL' direction will be axial (Z), the 'THICKNESS' direction will be radial (Y), and the 'SPAN' direction will be tangential (X). For element types that generate faces, face 1 will be the face seen in the normal engine cross section (ZY). Optionally a Patran Neutral File (partname.pnf) and/or a CSTEM input deck (partname.cst) can be written.

1.6.1.2.8. ERROR MESSAGES

- THIS SESSION FILE WAS WRITTEN BY VERSION cccc
  THERE ARE NO GUARANTEES THAT IT CAN BE READ BY THIS VERSION
  ENTER 1 TO CONTINUE

This is an informational message. You have entered a session file that was written by a prior version of the airfoil mesh generator. Changes in the number of questions, and the interpretation of answers, may mean that this session file will have to be modified. Try it, but watch the responses generated by the program.

1.6.1.2.8. ERROR MESSAGES (CONTINUED)

iiiii PAIRS OF POINTS PER SECTION iiii SECTIONS

This is an informational message. This is what the program has read from the input file, if it is not what you expect, or just plain wrong, the program will probably not get to much further.

MAXIMUM OF iii CHORD STATIONS EXCEEDED

This is a fatal error message. The program has certain internal limits, the number of chord stations is one of them, you have requested too many chordwise elements for the type of element requested. Try again with fewer chordwise elements.

NUMBER PAIRS OF POINTS PER SECTION CHANGED TO iiii

This is an informational message. The input file contains variable numbers of points per section. The number of points in this section is not the same as the previous section.

SECTION ii UNWRAPPED CHORD xx.xxxx

This is an informational message. This is the length of the unwrapped chord for this section. This line follows the center of the airfoil section. This message will help you with weighted spacing, and is a good check on airfoil size.

SECTION II PROJECTED CHORD xx.xxxx

This is an informational message. This is the length of projected chord for this section. This line follows the centerline of the engine. This message will help you with weighted spacing, and is a good check on airfoil size.

WEIGHTING FACTOR OUT OF ORDER x.xxxxx

This is a correctable error message. The weighting factors must be in increasing order from 0. to 1. The function will ask you to reenter all of the factors.

ERROR IN CHORD INTERPOLATION

This is a geometrical error message. The function could not find a location on the chord line to match a weighting factor. The function will attempt to go on, but will probably not work. Try a different chord fitting option.

NO INTERSECTION FOUND, cccccccc STATION ii

This is a geometrical error message. The function could not intersect the outer (pressure or suction) surface with a line normal to the chord. The function will attempt to go on, but will probably not work. Try a different chord fitting option.

1.6.1.2.8. ERROR MESSAGES (CONTINUED)

NO INTERSECTION FOUND CORRECTING ccccccccc STATION ii ii

This is a geometrical error message. While generating a VANS mid-side node the function could not intersect the outer (pressure or suction) surface with a line normal to the chord. The function will attempt to go on, but will probably not work. Try a different chord fitting option.

NUMBER OF NODES iiiii LAST NODE iiiii
NUMBER OF ELEMENTS iiiii LAST ELEMENT iiiii

This is an informational message. This is the size of the model you have created.

NOT ENOUGH (ii) POINTS PER SECTION

This is a fatal error message. There are not enough points per section to continue.

MAXIMUM OF iii POINTS PER SECTION EXCEEDED

This is a fatal error message. The program has certain internal limits, the number of points per section is one of them.

NOT ENOUGH (ii) INPUT SECTIONS

This is a fatal error message. There are not enough sections to continue.

MAXIMUM OF iii INPUT SECTIONS EXCEEDED

This is a fatal error message. The program has certain internal limits, the number of sections is one of them. Try eliminating a few sections, or create your model in multiple pieces.

CANNOT CORRECT ll LEADING AND jj TRAILING EDGE POINTS
NOT ENOUGH (ii) POINTS PER SECTION

This is a fatal error message. There are not enough points per section to continue. Try using fewer leading/trailing edge points.

ERROR IN CHORD cccccccccc FIT, TYPE ii
(-1 TOO MANY POINTS, -2 NOT IN ORDER, -3,-4 SOLUTION)

This is a fatal error message. You have requested spline type chord interpolation. There is a mathematical reason that your section cannot be fit as a spline. Try linear interpolation.

ERROR READING GLASS MASTER FILE. IER iiiiii NUM iiiiii
DATR x.xxxxxxxe ii x.xxxxxxxe ii x.xxxxxxxe ii

This is a fatal error message. This is probably not a glass master file.

1.6.1.2.8. ERROR MESSAGES (CONTINUED)

cccccc EDGE THICKNESS CHANGED FROM xx.xxxx TO xx.xxxx
CHANGE IN CHORD xx.xxxx

This is an informational message. This is how much the edge correction changed the thickness and chord.

BAD cccc EQUATIONS ii
(-1 TOO MANY POINTS, -2 NOT IN ORDER, -3,-4 SOLUTION)

This is a correctable error message. The set of points entered for tip or root cutting could not be fit as a spline. Try again.

MAXIMUM OF iii PARTITIONS EXCEEDED

This is a fatal error message. The program has certain internal limits, the number of span partitions is one of them, you have requested too many spanwise elements for the type of element requested. Try again with fewer chordwise elements, or create multiple pieces.

ERROR IN SPAN cccccccc FIT, TYPE ii
(-1 TOO MANY POINTS, -2 NOT IN ORDER, -3,-4 SOLUTION)

This is a fatal error message. You have requested spline type span interpolation. There is a mathematical reason that your station cannot be fit as a spline. Try linear interpolation.

STATION ii UNWRAPPED SPAN xxx.xxxx

This is an informational message. This is the length of the unwrapped span for this station. This line follows the center of the airfoil station. This message will help you with weighted spacing, and is a good check on airfoil size.

STATION ii PROJECTED SPAN xxx.xxxx

This is an informational message. This is the length of the projected span for this station. This line follows the span axis. This message will help you with weighted spacing, and is a good check on airfoil size.

ERROR IN SPAN INTERPOLATION

This is a geometrical error message. The function could not find a location on the span line to match a weighting factor. The function will attempt to go on, but will probably not work. Try a different span fitting option.

1.6.1.2.8. ERROR MESSAGES (CONTINUED)

NO INTERSECTION FOUND, cccccccc PARTITION iii

This is a geometrical error message. The function could not intersect the outer (pressure or suction) surface with a line normal to the span. The function will attempt to go on, but will probably not work. Try a different span fitting option.

FOR PARTITION ii STATION ii OVERLAP IN ccccccccccc THICKNESS

This is a geometrical error message. The plate thicknesses requested are thicker than the airfoil.

AT PARTITION ii STATION iii THE AIRFOIL IS x.xxxxxx THICK

This is an informational message. The function is reporting the airfoil thickness for some correction option, it may be followed by:

WHICH IS LESS THAN THE SUM OF THE COATINGS x.xxxxxxxx

This is an informational message. The airfoil thickness is less than the coatings to be subtracted from it. This message may be followed by:

THE CORRECTION OPTION IS ii

This is a fatal error message. The function can not continue with this correction option.

1.6.1.2.9. EXAMPLE

Following is an example of running the Component Specific Airfoil Generator. Note: there are 4 elements thru the thickness and the middle two rows of elements are removed for chord locations 4 and 6. See Figure 1.6.1.2.9 for a plot of the airfoil generated.

EXAMPLE HOLLOW AIRFOIL

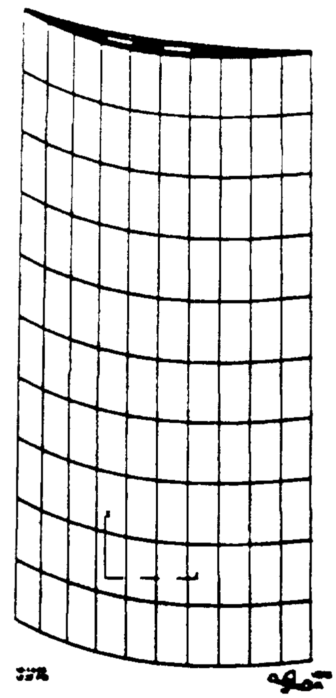


Figure 1.6.1.2.9

COSMO FUNCTION SUMMARY

FUNCTION:

Airfoil Mesh Generator

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 15 or **AIRFOIL**)

PURPOSE:

This function generates plate, brick or variable-noded solid element meshes from an aero coordinate definition. The output is a Unified Input File (UIF)

INPUT FILE(S):

Aero Geometry file (40)

OUTPUT FILE(S):

UIF (31)
Airfoil Paired Node UIF (32)

REQUIRED USER INPUT:

The chordwise, spanwise, and thickness mesh densities must also be specified as well as the element type to be generated

COMMENTS:

This function is particularly useful for the generation of 3D solid airfoils. (Fan and compressor vanes and blades). It also can be used to generate plate models of hollow stiffened struts

FUNCTION VERSION INFORMATION:

Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:

CSTEM Deck Generator

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 16 or **CP4**)

PURPOSE:

This function creates an input file for the CSTEM finite element analysis program from a Random Data Base (RDB).

INPUT FILE(S):

RDB (37)
CSTEM layer specification file (51)

OUTPUT FILE(S):

CSTEM deck (35)

REQUIRED USER INPUT:

The user must specify the IDNT data (either direct input or to approve using the ANLS CSTI data). Also for element layering, the user must specify the boundary surfaces to generate cross sections for.

COMMENTS:

Only CSTEM structural analysis is supported by this function.

FUNCTION VERSION INFORMATION:

Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:
Translate RDB to PATRAN Neutral File

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 14 or **PATRAN**)

PURPOSE:
This function creates an PATRAN Neutral File from a Random Data Base (RDB).

INPUT FILE(S):
RDB (37)

OUTPUT FILE(S):
PATRAN Neutral File (35)

REQUIRED USER INPUT:
An analysis title is requested. The analysis code type is requested. Node and element adders are requested. 2D models may be flipped from the SIESTA Y-Z plane to the PATRAN X-Y plane.

COMMENTS:
Not all options and all element types are currently supported. See the function description for details.

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:
Convert Random Data Base to an IGES Geometry File

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 13 or **GEOM**)

PURPOSE:
This function writes geometry information on a Random Data Base (RDB) to a IGES geometry file.

INPUT FILE(S):
RDB (37)

OUTPUT FILE(S):
IGES file (40)

REQUIRED USER INPUT:
The user must select the type of data to write to the IGES file and also input the identification data that IGES files require.

COMMENTS:
NONE

FUNCTION VERSION INFORMATION:
Any changes made to this function after version L01 05-94 will not be reflected in this release of the manual.

SIESTA FUNCTION SUMMARY

FUNCTION:
Bandwidth Optimizer (Gibbs-Poole-Stockmeyer Routine)

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 12 or **BAND**)

PURPOSE:
This routine attempts to reduce the nodal bandwidth of the data on a Random Data Base (RDB) using the GPS algorithm.

INPUT FILE(S):
RDB (37)

OUTPUT FILE(S):
The RDB may be modified to reflect the new node ordering.

REQUIRED USER INPUT:
After the banding routine is complete, you may reorder the RDB.

COMMENTS:
A new UIF, reflecting the new node ordering, may be written with the RDB to UIF function (Sub-Menu 2, Function 8 or **UIFWRITE**)

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:

2D to 3D Model Generation

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 11 or **TO3D**)

PURPOSE:

The purpose of this function is to generate a 3D UIF from 2D data on a Random Data Base (RDB). The 3D model may be generated by stacking along an axis, rotating about an axis, or filling between two planes.

INPUT FILE(S):

RDB (37)

OUTPUT FILE(S):

UIF (31)
Errors or warning messages (30)

REQUIRED USER INPUT:

You must specify the method for generating the 3D model (stack, rotate, fill), the number of element layers, and the spacing of the layers.

COMMENTS:

On-line information is available at any menu level.
This function currently supports these elements type conversions:

SHEL to PLAT
RING to BEAM
(2D) RIGI to (3D) RIGI
EL2D to BRI8
HT2D to HT3D
PE2D to VANS

FUNCTION VERSION INFORMATION:

Any changes made to this function after version L01 06-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:

Master Region Mesh Generator (MR.MESH)

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 10 or **MESH**)

PURPOSE:

This function generates a mesh of quadrilateral, triangular, or shell elements from a Random Data Base (RDB) containing master regions. This function will write a Unified Input File (UIF) with the mesh information, save the mesh information in the RDB, or both. Rigid conectors between axisymmetric shell elements and quadrilaterals are automatically generated. You may interactively modify the mesh, plot the model, or perform bandwidth reduction in this function.

INPUT FILE(S):

Random Data Base (37) containing master regions

OUTPUT FILE(S):

UIF (31) (on request)
RDB (37) (on request)

REQUIRED USER INPUT:

You must enter mesh definition information, if there is not enough information on the RDB to completely define the mesh.

COMMENTS:

The input RDB cannot contain any nodes or elements.

FUNCTION VERSION INFORMATION:

Any changes made to this function after version L01 05-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:

Translate a Patran Neutral File to a Unified Input File (UIF).

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 9 or **UF4**)

PURPOSE:

This function reads a PATRAN Neutral File and generates an equivalent UIF.

INPUT FILE(S):

PATRAN Neutral File

OUTPUT FILE(S):

UIF (31)

REQUIRED USER INPUT:

(1) The PATRAN Neutral File name.
(2) The analysis code corresponding to the PATRAN Neutral File.
(3) Whether the model should be flipped from the default PATRAN plane (XY) to the SIESTA plane (YZ) (for 2D).

COMMENTS:

Not all PATRAN data packet types are translated. Element property data cannot be processed if there is a discrepancy between the configuration number and analysis code entered.

FUNCTION VERSION INFORMATION:

Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:

Translate Random Data Base (RDB) to Unified Input File (UIF)

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 8 or **UIFWRITE**)

PURPOSE:

This function reads data from a RDB and writes a UIF. Nodes and elements may be renamed before they are written to the UIF.

INPUT FILE(S):

RDB (37)

OUTPUT FILE(S):

Initial UIF (47)
Subsequent UIFs using the NEWF command (56 thru 60)

REQUIRED USER INPUT:

You must specify which data to retrieve. You may also rename items on the output UIF.

COMMENTS:

This function can selectively retrieve:

1) All RDB input data
2) Geometric data
3) Boundary Condition data
4) Specific input data defined by primary and secondary keys

FUNCTION VERSION INFORMATION:

Any changes made to this function after version 1.02 06-94 will not be reflected in this release of the manual.

SIESTA FUNCTION SUMMARY

FUNCTION:
SIESTA 2D Plotting

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 7 or **PLOT**)

PURPOSE:
This routine generates plots from a Unified Plot File (UPF) . Copies of the plots may be directed to Postscript plotters.

INPUT FILE(S):
UPF (51)

OUTPUT FILE(S):
Postscript plot file (36)

REQUIRED USER INPUT:
The plot specifications may be entered interactively

COMMENTS:
On-line help is available

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.02 06-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:
SIESTA Graphics

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 6 or **GRAPHICS**)

PURPOSE:
This function generates plots from a Random Data Base (RDB) . Various labelling options as well as hidden line, free edge, displaced and contour plots are available. Copies of the plots may be directed to plotters.

INPUT FILE(S):
RDB (37)

OUTPUT FILE(S):
PLOT FILE (36)

REQUIRED USER INPUT:
The plot specifications must be entered interactively.

COMMENTS:
On-line help is available. Window information may be saved for use other functions (e.g. RDB Extract function.)

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.05 06-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:
NASTRAN Deck Generator

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 5 or **NASTRAN**)

PURPOSE:
This function creates a NASTRAN Bulk Data Deck from a Random Data Base (RDB).

INPUT FILE(S):
RDB (37)

OUTPUT FILE(S):
NASTRAN bulk data deck (35)

REQUIRED USER INPUT:
NONE

COMMENTS:
NONE

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:

Edit Random Data Base

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 4 or **EDIT**)

PURPOSE:

This function permits you to modify an existing Random Data Base (RDB) with data on an auxiliary UIF. You may add additional information or modify (overwrite) current data.

INPUT FILE(S):

RDB (37)
UIF containing change information. (32)

OUTPUT FILE(S):

The RDB is modified to reflect the changes.

REQUIRED USER INPUT:

None, unless you have specified terminal input. Certain error conditions may request input from the terminal.

COMMENTS:

New material definitions may be added; however, current material data may not be altered.

FUNCTION VERSION INFORMATION:

Any changes made to this function after version L01 05-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:
Random Data Base Surface Generator

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 3 or **SURFACE**)

PURPOSE:
The function identifies the free surfaces and free edges of all 3D solid elements (BRI8, VANS, and HT3D) and writes this information to the Random Data Base (RDB) . In addition, this function can be used to tag "boundary surfaces" for boundary condition application. You must run this function prior to making hidden-line or free-edge plots via the SIESTA Graphics function (Sub-Menu 2, Function 6 or **GRAPHICS**).

INPUT FILE(S):
RDB (37)

OUTPUT FILE(S):
The RDB is updated to contain the free-edge and surface information, and, optionally, boundary surfaces.

REQUIRED USER INPUT:
If you choose to tag boundary surfaces, additional input may be needed to identify the desired surfaces.

COMMENTS:
NONE

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:
Random Data Base Outline Generator

SUB-MENU LOCATION:
SIESTA Functions (Sub-Menu 2, Function 2 or **OUTLINE**)

PURPOSE:
This function determines the free edges and connected outlines of regions in the Random Data Base (RDB) comprised of EL2D, PE2D or HT2D elements and stores this information on the RDB. This permits you to check for "cracks" or "holes" in a discretized area by plotting this outline and to assign boundary conditions using the outline. In addition, this function can be used to tag "boundary surfaces" for boundary condition application.

INPUT FILE(S):
RDB (37)

OUTPUT FILE(S):
The RDB is updated to contain free edge and outline information, and, optionally, boundary surfaces.

REQUIRED USER INPUT:
None for outline only. User input may be required to define boundary surfaces.

COMMENTS:
NONE

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.00 04-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:

Create SIESTA Random Data Base (RDB) from Unified Input File (UIF)

SUB-MENU LOCATION:

SIESTA Functions (Sub-Menu 2, Function 1 or **UIFREAD**)

PURPOSE:

This function reads a Unified Input File (UIF), checks the syntax, and writes a Random Data Base (RDB).

INPUT FILE(S):

Unified Input File (31)

OUTPUT FILE(S):

RDB (37)

REQUIRED USER INPUT:

None unless terminal input is requested in the UIF or certain error conditions are encountered.

COMMENTS:

The default values of maximum node and element name are 20480. The default values of maximum master node and master region name are 4096. The default values of maximum geometry node and geometry entity name are 10240. These may be reset as described in this description.

FUNCTION VERSION INFORMATION:

Any changes made to this function after version I.01 05-94 will not be reflected in this release of the manual.

1.6.1.5.1. FUNCTION DESCRIPTION

This function permits you to ·ew the COSMO News messages. Currently there are five News function options. The options are:

1) COSMO Driver Message
2) COSMO Phone List
4) COSMO News Messages

Any or all of the messages can be printed to the screen by entering the option numbers. The News Messages are written in chronological order starting with the most recent message. The News messages will be updated to tell the users of any new functions or enhancements to current functions. The Driver message will contain the date of the last News message update. Enter Q or QUIT to exit the NEWS function.

COSMO FUNCTION SUMMARY

FUNCTION:
COSMO News Function

SUB-MENU LOCATION:
COSMO Geometry (Sub-Menu 1, Function 5 or **NEWS**)

PURPOSE:
This function prints out the COSMO Driver Message, Phone List, and NEWS.

INPUT FILE(S):
NONE

OUTPUT FILE(S):
NONE

REQUIRED USER INPUT:
You must select the message that you want printed to the screen.

COMMENTS:

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.00 06-94 will not be reflected in this release of the manual.

COSMO FUNCTION SUMMARY

FUNCTION:
Print COSMO Menu Structure

SUB-MENU LOCATION:
COSMO Geometry (Sub-Menu 1, Function 4 or **MENU**)

PURPOSE:
This function prints out the current COSMO menu structure showing all sub-menus and functions. The entire COSMO menu structure is written to file 56.

INPUT FILE(S):
NONE

OUTPUT FILE(S):
COSMO menu file (56)

REQUIRED USER INPUT:
At the [MORE] prompt, enter carriage return to continue printing the COSMO menu or enter "Q" to stop printing the COSMO menu to the terminal.

COMMENTS:
NONE

FUNCTION VERSION INFORMATION:
NONE

```
     ENTER 1 TO GENERATE A PATRAN NEUTRAL FILE
1
     ##############################################################
         SIESTA PATRAN INPUT GENERATOR     VERS I.00  04-11-94
     ##############################################################
 DATA BASE VERS I.00  04-11-94
     ENTER THE PATRAN NEUTRAL FILE TITLE (MAX 40 CHARACTERS)
THIS IS A TEST FOR DISKS
     ENTER THE ANALYSIS CODE:
     1-ANSYS  2-NASTRAN  3-SIESTA  4-PTHERMAL  5-UNIGRAPHICS
2
 $
 $   TOTAL DATA BASE CONTENTS
 $            FIRST     LAST      MIN      MAX  NUMBER
 $    NODES       2     1656        1     1675    1230
 $    ELEMS       0        0        1      800     800
 $    BRI8S       1      800        1      800     800
 $
     WRITING PATRAN NEUTRAL FILE
          500 NODES WRITTEN
         1000 NODES WRITTEN
         1230 TOTAL NODES WRITTEN
          492 SETS OF NODAL DISPLACEMENTS WRITTEN
          200 BRI8S WRITTEN
          400 BRI8S WRITTEN
          600 BRI8S WRITTEN
          800 BRI8S WRITTEN
          800 TOTAL BRI8S WRITTEN

     YOUR PATRAN NEUTRAL FILE IS ON FILE  35
     THE COSMO DISK SHELL UIF IS part1.pnf
     ENTER 1 TO GENERATE A CSTEM INPUT FILE
1
     ##############################################################
          SIESTA CSTEM DECK GENERATOR  VERS I.00  04-11-94
     ##############################################################
 DATA BASE VERS I.00  04-11-94
          TOTAL DATA BASE CONTENTS
        DATA TYPE          MIN      MAX     TOTAL
     -----------------   ------   ------   ------
           NODE              1     1675     1230
           BRI8              1      800      800

     ENTER THE ANALYSIS IDENTIFICATION (UP TO 80 CHAR.)
THIS IS A TEST FOR DISKS
          500 NODES WRITTEN
         1000 NODES WRITTEN
         1230 NODES WRITTEN
          200 8-NODED BRICKS WRITTEN
          400 8-NODED BRICKS WRITTEN
          600 8-NODED BRICKS WRITTEN
          800 8-NODED BRICKS WRITTEN
          492 NODAL ANGLE LINES WERE WRITTEN
          492 NODAL FIXITY LINES WERE WRITTEN
     YOUR CSTEM DECK IS ON FILE 35
     THE MODEL CSTEM INPUT FILE IS part1.cst

          YOU ARE IN THE COSMO GEOMETRY SUB-MENU (1)
     ENTER DESIRED FUNCTION BY NUMBER OR ?,G,Q,SYSTEM
Q

     ##################################################################
       IT IS NOW  2:02 PM ON 06-30-94 WE THANK YOU FOR YOUR PATRONAGE.
     ##################################################################
```

```
    ENTER GENERATION OPTION
          (Q) - EXIT FROM 2D TO 3D, (?) - MENU, (INFO) - INFORMATION
          (0) ROTATE A 2D MODEL ABOUT AN AXIS
          (1) STACK A 2D MODEL ALONG AN AXIS
          (2) FILL BETWEEN TWO SURFACES
     0
    YOUR MODEL HAS COORDINATES IN ALL THREE AXES.
    ENTER COORDINATE DESCRIPTION OPTION
          (Q) EXIT 2D TO 3D
          (0) THE MODEL IS IN THE YZ PLANE
          (1) THE MODEL IS IN THE XZ PLANE
          (2) THE MODEL IS IN THE XY PLANE
     0
    ENTER DESIRED ROTATION OPTION
          (Q) - EXIT FROM 2D TO 3D, (?) - MENU, (INFO) - INFORMATION
          (0) EQUALLY SPACED ROTATION
          (1) UNEQUALLY SPACED ROTATION
          (INFO) FOR MORE INFORMATION
     0
    ENTER DESIRED SYMMETRY BOUNDARY CONDITION OPTION
          (0) APPLY NO BOUNDARY CONDITIONS
          (1) APPLY SYMMETRIC BOUNDARY CONDITIONS
          (2) APPLY ANTI-SYMMETRIC BOUNDARY CONDITIONS
          (3) APPLY PSEUDO SYMMETRIC BOUNDARY CONDITIONS
     1
    YOU HAVE SELECTED EQUALLY SPACED ROTATION.
    ENTER THE AXIS YOU WISH TO ROTATE ABOUT AS Y OR Z
z
    ENTER THE NUMBER OF ELEMENT LAYERS, THE ANGLE BETWEEN LAYERS,
          THE ANGLE OF THE PARENT LAYER, THE ANGLE OF TWIST,
          THE NODE NAME ADDER, AND THE ELEMENT NAME ADDER
4. 2.5
                   200 NODES PROCESSED
                   400 NODES PROCESSED
                   600 NODES PROCESSED
                   800 NODES PROCESSED
                  1000 NODES PROCESSED
                  1230 NODES PROCESSED
    GENERATING SYMMETRY BOUNDARY CONDITIONS
           0 NODAL ZERO DISPLACEMENTS PROPAGATED
              200 EL2DS PROCESSED INTO      200 BRI8S
              200 EL2DS PROCESSED INTO      400 BRI8S
              200 EL2DS PROCESSED INTO      600 BRI8S
              200 EL2DS PROCESSED INTO      800 BRI8S
    YOUR NEW 3D UIF IS ON FILE 31
    ################################################################
        UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
    ################################################################
STORAGE   VERS I.01   5-19-94
DATA BASE VERS I.00  04-11-94
READER    VERS I.04  06-02-94

    INPUT FILE HAS BEEN PROCESSED
    YOUR RANDOM DATA BASE IS ON FILE  37

    THE DISK 3D MODEL FILE IS part1.3uf

DATA BASE VERS I.00  04-11-94

    THE 3D DISK MODEL WEIGHT IS    .6288     LB
    THE TOTAL DISK WEIGHT IS       22.64     LB
```

```
PRIMARY KEY SET TO NODE
SECONDARY KEYS RESET TO NAME X     Y     Z     REFJ EXTJ ANAM NSEC
  TEMP THK  NODP UDAT
200 LINES WRITTEN
243 LINES WRITTEN
  SECONDARY KEYS RESET TO NAME ANG1 ANG2 ANG3 PW   PIXX PIYY PIZZ
  PMR
200 LINES WRITTEN
247 LINES WRITTEN
  SECONDARY KEYS RESET TO NAME GN01 GN02 GN03 GN04 GN05 GN06 GN07
  GN08 GN09 GN10 GN11 GN12
  0 LINES WRITTEN
  .     .     .     .     .
  .     .     .     .     .

  PRIMARY KEY SET TO EL2D
  SECONDARY KEYS RESET TO NAME CONN THK  TYPE GNUM SUBS OUTC MATL
    TI   TJ   TK   TL
200 LINES WRITTEN
202 LINES WRITTEN
  .     .     .     .     .
  .     .     .     .     .
THERE IS NO CURRENT PRIMARY KEY
    THE UIF IS ON FILE 47
    SELECT DESIRED OPTION
         -1 - QUIT (DO NOT SAVE MESH)
          0 - PLOT MESH
          1 - BANDWIDTH REDUCTION
          2 - WRITE UIF
          3 - DEFINE/MODIFY MESH
          4 - GENERATE MESH
         10 - QUIT (SAVE MESH ON RDB)
-1
DATA BASE VERS I.00  04-11-94
    THE COSMO DISK 2D MESH UIF IS part1.2uf
    THE COSMO DISK SHELL UIF IS  part1.suf

    ENTER THE NUMBER OF BLADES AND THE NUMBER OF
    CIRCUMFERENTIAL ELEMENTS TO USE
36 4
    ENTER THE   3 CIRCUMFERENTIAL BIASING PARAMETERS
    ENTER AS PERCENTS, THE SUM BEING LESS THAN 1.0
    ENTER CARRIAGE RETURN FOR EQUAL SPACING
<CR>
    ##############################################################
        UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
    ##############################################################
STORAGE   VERS I.01   5-19-94
DATA BASE VERS I.00  04-11-94
READER    VERS I.04  06-02-94

    INPUT FILE HAS BEEN PROCESSED
    YOUR RANDOM DATA BASE IS ON FILE  37
    ##############################################################
        SIESTA 2D TO 3D MODEL GENERATOR  VERS I.01  06-23-94
    ##############################################################
DATA BASE VERS I.00  04-11-94
    ENTER STRUCTURAL MODEL PROPAGATION OPTION
         (Q) - EXIT FROM 2D TO 3D, (?) - MENU, (INFO) - INFORMATION
         (0)  DO NOT PROPAGATE TEMPERATURES, MATERIAL CODES
         (1)  PROPAGATE TEMPERATURES FROM 2D TO 3D MODEL
         (2)  PROPAGATE MATERIAL CODES FROM 2D TO 3D MODEL
         (3)  PROPAGATE BOUNDARY SURFACES FROM 2D TO 3D MODEL
         (4)  SHIFT PARENT LAYER NODE AND ELEMENT NAMES
         (5)  DISTRIBUTE POINT WEIGHTS OVER THE ROTATED MODEL
2. 3.
```

```
          4 - GENERATE MESH
         10 - QUIT (SAVE MESH ON RDB)
4
 STORAGE   VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
     THERE ARE STILL      64 UNDEFINED EDGES
     HOW WOULD YOU LIKE TO DEFINE THE MESH ON THESE EDGES
          0 - ASPECT RATIO
          1 - ELEMENT SIZE OR ANGLE
          2 - CONSTANT NUMBER OF ELEMENTS
1
     ENTER MAXIMUM ELEMENT SIZE AND MAXIMUM ARC ANGLE
.1
     ESTIMATED MODEL SIZE
             336 NODES,
             200 ELEMENTS

     DO YOU WANT TO SKIP GENERATION? (Y/N)
<CR>
     ENTER NODE AND ELEMENT NUMBER OFFSETS
     (OR RETURN TO START WITH NODE 1 AND ELEMENT 1)
<CR>
     MESHING ISLAND       1
          MESHING REGION       1
          MESHING REGION       2
          MESHING REGION       3
          MESHING REGION       4
          MESHING REGION       5
          MESHING REGION       6
          MESHING REGION       7
          MESHING REGION       8
          MESHING REGION       9
          MESHING REGION      10
          MESHING REGION      11
          MESHING REGION      12
          MESHING REGION      13
          MESHING REGION      14
          MESHING REGION      15
          MESHING REGION      16
     ACTUAL MODEL SIZE
             246 NODES,
             200 ELEMENTS

     SELECT DESIRED OPTION
         -1 - QUIT (DO NOT SAVE MESH)
          0 - PLOT MESH
          1 - BANDWIDTH REDUCTION
          2 - WRITE UIF
          3 - DEFINE/MODIFY MESH
          4 - GENERATE MESH
         10 - QUIT (SAVE MESH ON RDB)
2
     ##############################################################
             SIESTA RDB TO UIF WRITER VERS I.02  06-20-94
     ##############################################################
 DATA BASE VERS I.00  04-11-94
 FULL DUMP OF RDB

     PRIMARY KEY SET TO NLIM
     SECONDARY KEYS RESET TO BNOD LNOD BELM LELM BMND LMND BMRG LMRG
       BGND LGND BGEO LGEO
     0 LINES WRITTEN
     SECONDARY KEYS RESET TO BLST LLST BSTD LSTD FLSZ
     0 LINES WRITTEN

     PRIMARY KEY SET TO NGDN
     0 LINES WRITTEN
```

```
          CODE        VALUE                        CODE        VALUE
            1        10.00000                        2        10.00000
            3        14.00000                        4          .50000
            5          .50000                        6          .40000
            7          .40000                        8          .50000
            9          .25000                       10          .10000
           11          .15000                       12          .20000
           13          .40000                       14        -1.00000
           15        -1.00000                       16        -1.00000

     ENTER PARAMETER CHANGES (ENTRY CODE, NEW VALUE)
     OR <CR> TO GENERATE THE DISK MODEL GEOMETRY
     OR "FILE" TO ENTER PARAMETERS FROM A FILE
     OR "LIST" TO LIST OF THE PARAMETER VALUES
     OR "QUIT" TO QUIT
<CR>
     THE DISK MODEL PARAMETER FILE IS part1.par
     ENTER 1 TO GENERATE AN IGES FILE
1
     ##########################################################
         UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
     ##########################################################
 STORAGE   VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
 READER    VERS I.04  06-02-94

     INPUT FILE HAS BEEN PROCESSED
     YOUR RANDOM DATA BASE IS ON FILE  37
     ##########################################################
           SIESTA GEOMETRY RECREATOR    VERS I.01  05-05-94
     ##########################################################
 DATA BASE VERS I.00  04-11-94
     GEOM DATA WILL BE CONVERTED
     ENTER THE PART NAME (UP TO 10 CHAR.):
DISK
     ENTER YOUR NAME (UP TO 20 CHAR.):
COSMO TEST
     ENTER YOUR ORGANIZATION NAME (UP TO 20 CHAR.):
GEAE
           16 GENTS WRITTEN
     THE IGES GEOMETRY DATA IS ON FILE  40
     THE COSMO DISK IGES FILE IS part1.igs
     ##########################################################
         UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
     ##########################################################
 STORAGE   VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
 READER    VERS I.04  06-02-94

     INPUT FILE HAS BEEN PROCESSED
     YOUR RANDOM DATA BASE IS ON FILE  37
     ##########################################################
     MASTER REGION MESH GENERATOR (MR. MESH)  VERS I.01  05-09-94
     ##########################################################
 STORAGE   VERS I.01   5-19-94
 DATA BASE VERS I.00  04-11-94
     THERE ARE:
               42 MASTER NODES
               16 MASTER REGIONS
               17 STRIPS
                1 MESH ISLANDS
     SELECT DESIRED OPTION
          -1 - QUIT (DO NOT SAVE MESH)
           0 - PLOT MESH
           1 - BANDWIDTH REDUCTION
           2 - WRITE UIF
           3 - DEFINE/MODIFY MESH
```

Component Specific Airfoil Generator Example (Input is underlined)

```
> xcosmo
    ####################################################################
                 CCCCC    OOOOOO    SSSSSS   MM    MM   OOOOOO
                 CC       OO  OO    SS       MMM MMM    OO  OO
                 CC       OO  OO    SSSSSS   MM M MM    OO  OO
                 CC       OO  OO        SS   MM M MM    OO  OO
                 CCCCCC   OOOOOO    SSSSSS   MM    MM   OOOOOO
    IT IS  4:27 PM ON 06-30-94 SYSTEM c0401        VERS I.01 06-30-94
    ####################################################################
     This is Version I.01 of COSMO (Production).
     Type NEWS for the latest COSMO News - last update 06-29-94.
          Type NEWS for more information.

                    AVAILABLE SUB-MENUS
    (?)  MENU, (Q)  QUIT, (SYSTEM) SYSTEM
     0 EXIT FROM COSMO
     1 COSMO GEOMETRY                   2 SIESTA FUNCTIONS
    31 CUSTOMIZE COSMO

         CURRENT MAXIMUM SUB-MENU IS 31

         YOU ARE IN THE MAIN MENU (0)
    ENTER DESIRED SUB-MENU BY NUMBER OR ?,G,Q,SYSTEM
AGEN
    ##############################################################
      COMPONENT SPECIFIC AIRFOIL GENERATOR VERS I.02  06-30-94
              ON c0401         AT  4:27 PM 06-30-94
    ##############################################################

    ENTER THE PART NAME (UP TO 8 CHARACTERS)
part1
    ENTER THE AIRFOIL FILENAME
input.af
    ENTER THE NUMBER OF PAIRS OF POINTS PER SECTION
39
    ENTER THE NUMBER OF AIRFOIL SECTIONS
2
    ENTER 1 TO MODIFY THE INPUT DATA

    ENTER THE NUMBER OF ELEMENTS ALONG THE CHORD
10
    ENTER 0 FOR EVEN CHORD SPACING
          3 FOR USER DEFINED SPACING
0
    ENTER THE NUMBER OF ELEMENTS ALONG THE SPAN
10
    ENTER THE NUMBER OF ELEMENTS ACROSS THE THICKNESS
4
    ENTER 0 FOR THE EQUALLY SPACED METHOD
          1 FOR THE MULITPLE COATING (SKIN) METHOD
0
    ENTER THE ELEMENT CAVITY THICKNESS LAYER NUMBERS
    ENTER CARRIAGE RETURN FOR A SOLID AIRFOIL
2 3
    ENTER THE ELEMENT CAVITY CHORD ROW NUMBERS
4 6
    ##############################################################
      SIESTA AIRFOIL MODELING AIDS PROGRAM VERS I.00  04-11-94
    ##############################################################
    ENTER 0 IF THE FILE IS AN AERO COORDINATE FILE
          1 IF THE FILE CONTAINS DIGITIZED DATA
          2 IF THE FILE IS AN AIG SHROUD FILE
          3 IF THE FILE IS AN AIG AIRFOIL FILE
          4 IF THE FILE IS AN AMP AIRFOIL FILE
 1
    ENTER THE NUMBER OF PAIRS OF POINTS PER SECTION
```

```
39
    ENTER THE NUMBER OF AIRFOIL SECTIONS
2
    ENTER 1 TO MODIFY THE INPUT DATA
0
    ENTER THE NUMBER OF NODES PER ELEMENT (-4,0,3,4,8,9,16,20)
20
    ENTER THE NUMBER OF ELEMENTS ALONG THE CHORD
10
    ENTER 0 FOR EVEN CHORD SPACING
          1 FOR POWER SPACING
          2 FOR BI-POWER SPACING
          3 FOR USER DEFINED SPACING
0
DATA BASE VERS I.00  04-11-94
    SECTION  1 UNWRAPPED CHORD  3.4178
    SECTION  2 UNWRAPPED CHORD  3.4910
    ENTER 0 TO NOT CHANGE THE  TIP FLOWPATH CONTOUR
          1 TO DEFINE A PLANAR FLOWPATH CONTOUR
          2 TO DEFINE A CONICAL FLOWPATH CONTOUR
0
    ENTER 0 TO NOT CHANGE THE ROOT FLOWPATH CONTOUR
          1 TO DEFINE A PLANAR FLOWPATH CONTOUR
          2 TO DEFINE A CONICAL FLOWPATH CONTOUR
0
    ENTER THE NUMBER OF ELEMENTS ALONG THE SPAN
10
    ENTER 0 FOR EVEN SPAN SPACING
          1 FOR POWER SPACING
          2 FOR BI-POWER SPACING
          3 FOR USER DEFINED SPACING
0
    STATION  1 UNWRAPPED SPAN 10.0001
    STATION  2 UNWRAPPED SPAN 10.0000
    STATION  3 UNWRAPPED SPAN 10.0000
    STATION  4 UNWRAPPED SPAN 10.0000
    STATION  5 UNWRAPPED SPAN 10.0000
    STATION  6 UNWRAPPED SPAN 10.0000
    STATION  7 UNWRAPPED SPAN 10.0000
    STATION  8 UNWRAPPED SPAN 10.0000
    STATION  9 UNWRAPPED SPAN 10.0000
    STATION 10 UNWRAPPED SPAN 10.0000
    STATION 11 UNWRAPPED SPAN 10.0000
    STATION 12 UNWRAPPED SPAN 10.0000
    STATION 13 UNWRAPPED SPAN 10.0000
    STATION 14 UNWRAPPED SPAN 10.0000
    STATION 15 UNWRAPPED SPAN 10.0000
    STATION 16 UNWRAPPED SPAN 10.0001
    STATION 17 UNWRAPPED SPAN 10.0001
    STATION 18 UNWRAPPED SPAN 10.0001
    STATION 19 UNWRAPPED SPAN 10.0001
    STATION 20 UNWRAPPED SPAN 10.0002
    STATION 21 UNWRAPPED SPAN 10.0002
    ENTER THE NUMBER OF ELEMENTS ACROSS THE THICKNESS
-4
    ENTER 0 FOR THE EQUALLY SPACED METHOD
          1 FOR THE MULITPLE COATING (SKIN) METHOD
0
    ENTER THE ELEMENT CAVITY THICKNESS LAYER NUMBERS
    ENTER CARRIAGE RETURN FOR A SOLID AIRFOIL
2 3
    ENTER THE ELEMENT CAVITY CHORD ROW NUMBERS
4 6
    ENTER FIRST NODE NUMBER, FIRST ELEMENT NUMBER,
          SPAN NODE NUMBER ADDER, AND SPAN ELEMENT NUMBER ADDER
1 1
    NUMBER OF NODES          2189 LAST NODE          2189
```

```
    NUMBER OF ELEMENTS      360 LAST ELEMENT      400
    NUMBER OF BOUNDARY SURFACES        6
    NUMBER OF AIRFOIL PAIRED NODE STRUTS      121 WRITTEN TO FILE  32
    ENTER 1 FOR FIXED TIP  BOUNDARY CONDITIONS
0
    ENTER 1 FOR FIXED ROOT BOUNDARY CONDITIONS
0
    YOUR UIF IS ON FILE 31
    ##############################################################
        UNIFIED INPUT FILE (UIF) READER  VERS I.01  05-25-94
    ##############################################################
STORAGE   VERS I.01   5-19-94
DATA BASE VERS I.00  04-11-94
READER    VERS I.04  06-02-94

    INPUT FILE HAS BEEN PROCESSED
    YOUR RANDOM DATA BASE IS ON FILE  37

    THE AIRFOIL MODEL FILE IS part1.3uf

    ENTER 1 TO GENERATE A PATRAN NEUTRAL FILE
1
    ##############################################################
        SIESTA PATRAN INPUT GENERATOR    VERS I.00  06-28-94
    ##############################################################
DATA BASE VERS I.00  04-11-94
    ENTER THE PATRAN NEUTRAL FILE TITLE (MAX 40 CHARACTERS)
COSMO AIRFOIL
    ENTER THE ANALYSIS CODE:
    1-ANSYS  2-NASTRAN  3-SIESTA  4-PTHERMAL  5-UNIGRAPHICS
1
$
$   TOTAL DATA BASE CONTENTS
$            FIRST     LAST      MIN      MAX  NUMBER
$    NODES       1     2189        1     2189    2189
$    ELEMS       0        0        1      400     360
$    VANSS       1      400        1      400     360
$
    WRITING PATRAN NEUTRAL FILE
         500 NODES WRITTEN
        1000 NODES WRITTEN
        1500 NODES WRITTEN
        2000 NODES WRITTEN
        2189 TOTAL NODES WRITTEN
         360 TOTAL VANS WRITTEN
           0 VANS FACE NORMAL PRESSURES WRITTEN

    YOUR PATRAN NEUTRAL FILE IS ON FILE  35
    THE MODEL PATRAN NEUTRAL FILE IS part1.pnf

    ENTER 1 TO GENERATE A CSTEM INPUT FILE
1
    ##############################################################
         SIESTA CSTEM DECK GENERATOR  VERS I.00  04-11-94
    ##############################################################
DATA BASE VERS I.00  04-11-94
          TOTAL DATA BASE CONTENTS
         DATA TYPE          MIN      MAX     TOTAL
    ------------------   ------   ------   ------
            NODE              1     2189     2189
            VANS              1      400      360

    ENTER THE ANALYSIS IDENTIFICATION (UP TO 80 CHAR.)
COSMO AIRFOIL
         500 NODES WRITTEN
        1000 NODES WRITTEN
        1500 NODES WRITTEN
```

```
       2000 NODES WRITTEN
       2189 NODES WRITTEN
     360 20-NODED BRICKS WRITTEN
   ENTER THE LAYER SPECIFICATION DATA FILE NAME
   THIS DATA WILL BE ADDED TO THE CSTEM DECK
   OR ENTER 'NONE' TO SKIP ELEMENT LAYERING
NONE
   YOUR CSTEM DECK IS ON FILE 35
   THE MODEL CSTEM INPUT FILE IS part1.cst

        YOU ARE IN THE MAIN MENU (0)
   ENTER DESIRED SUB-MENU BY NUMBER OR ?,G,Q,SYSTEM
q

   ##########################################################################
     IT IS NOW  4:28 PM ON 06-30-94 WE THANK YOU FOR YOUR PATRONAGE.
   ##########################################################################
```

COSMO FUNCTION SUMMARY

FUNCTION:
Disk Model Generator

SUB-MENU LOCATION:
COSMO Geometry (Sub-Menu 1, Function 3 or **DISK**)

PURPOSE:
This function generates 2D and 3D disk models using a set of parameters to define the disk cross section. The output is a Unified Input File (UIF). Optionally an IGES file, Patran Neutral File and/or a CSTEM input deck can be generated.

INPUT FILE(S):
Disk Parameter File (52)

OUTPUT FILE(S):
Disk Parameter File (partname.par)
2D IGES file (partname.igs)
2D Mesh UIF (partname.2uf)
2D Shell UIF (partname.suf)
3D Disk UIF (partname.3uf)
3D Patran Neutral File (partname.pnf)
3D CSTEM input file (partname.cst)

REQUIRED USER INPUT:
The part name used to name output files must be input. The disk parameters, number of blades, number of circumferential elements and element circumferential spacing must also be specified.

COMMENTS:
NONE

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 1.01 06-94 will not be reflected in this release of the manual.

1.6.1.3.1. INTRODUCTION

The disk model generator generates a finite element mesh of a disk using a set of specified parameters to define the disk cross section. In the future, certain parameters can be read from a T/BEST Neutral File. The output is a Unified Input File (UIF). Optionally a 2D mesh and geometry UIF, a 2D Shell UIF, a 2D IGES file, a Patran Neutral File and/or a CSTEM input deck can be generated. The 3D mesh will consist of 8-nodel solid elements.

When you run this function, you are prompted for the part name. This part name is used to name output files in COSMO. A part name may be up to eight characters. If the part name is entered as partname, then the 3D UIF would be partname.3uf.

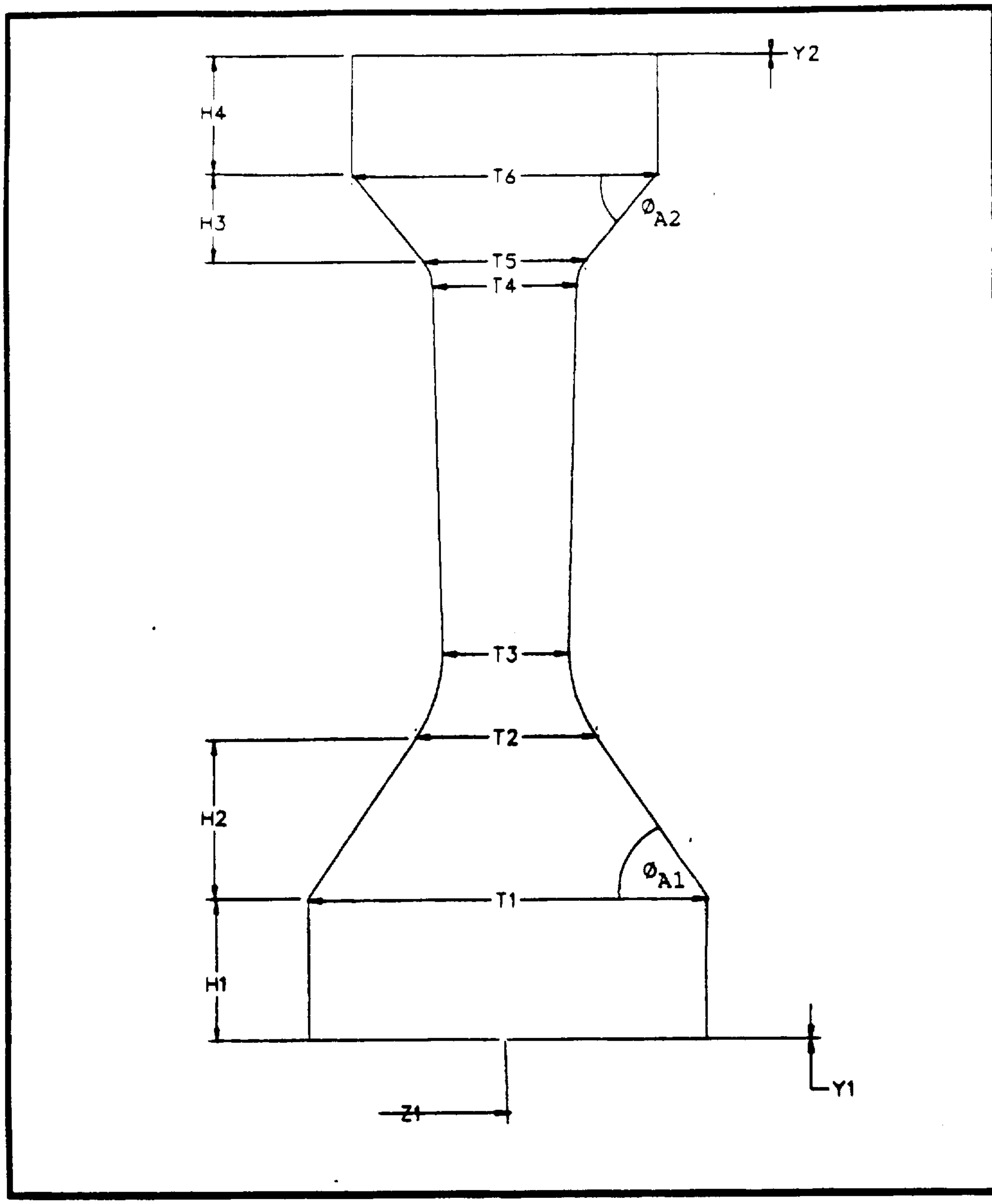
1.6.1.3.2. DISK PARAMETERS

The disk cross section is defined by 13 geometric parameters. Figure 1.6.1.3.1 shows the disk cross section and parameters. The parameters can be input interactively or using an input file. In the future, certain parameters can be set by the T/BEST Neutral File.

There are also three control parameters:

14	IGES Write
15	Patran Neutral File Write
16	CSTEM Deck Write

The possible values for the control parameters are: -1 prompt user for the process, 0 skip the process, or 1 perform the process. The default values of these parameters is -1.



Code	Name	Code	Name
1	Z1	2	Y1
3	Y2	4	H1
5	H2	6	H3
7	H4	8	T1
9	T2	10	T3
11	T4	12	T5
13	T6		

Figure 1.6.1.3.1
COSMO Disk Model Parameters
Z = Z coordinate, Y = Y coordinate, H = Height, T = Thickness

1.6.1.3.3. DISK GEOMETRY CHECKS

Once the disk parameters have been specified, the disk geometry is checked to make sure that a valid disk geometry can be generated from the parameters. There are several different checks:

All geometric parameters (2-13) must be greater than 0. (Z1 can be 0.)
Y2 must be greater than Y1 for a positive disk height
T1 must be greater than T2
T2 must be greater than T3
T5 must be greater than T4
T6 must be greater than T5
Y2-Y1 = disk height must be greater than the sum of the heights H1+H2+H3+H4
$\varnothing_{A1}$ and $\varnothing_{A2}$ must be less than 60° (see figure 1.6.1.3.1)

The angles checks are needed to ensure that a radius can be generated between the bore and the web and also between the rim and the web. If any of these geometry checks fail, an error message is written to the screen and you are prompted to enter new disk parameters.

1.6.1.3.4. DISK MODEL GENERATION

Once the a valid disk geometry has been checked, the 2D cross section model is generated. You are prompted to enter 1 to write out the 2D IGES file. If you select this option, you will be asked for the part name, your name, and your organization. This data is written to the IGES file. The IGES file is written as partname.igs. Next, a 2D mapped mesh is created for the cross section. Currently, you have to answer several questions to generate the 2D mesh. In the future, this will be automated.

The Master Region Mesh Generator (Sub-Menu 2, Function 10 or MESH) generates a 2D mapped mesh from a master region definition. The details of mesh generation and master regions are in the SIESTA manual. To generate the 2D mesh, enter the following commands:

```
  4     $ Generate Mesh
  1     $ Set the Element Size for the mesh
  .1    $ Element Size set to 0.1 in
 <CR>   $ Do not skip mesh generation
 <CR>   $ Use the default node and element numbering
  2     $ Write the Mesh UIF
  -1    $ Quit and return to Disk Model Generator.
```

The 2D Mesh UIF is written as partname.2uf. A separate 2D Shell UIF is written as partname.suf. This UIF could be used for a Shell model of the disk or as input to a disk analysis program.

The 2D mesh is then rotated into a 3D sector model of the disk. The 3D model is a symmetry model of one blade. The model is rotated about the Z axis. You will be prompted for the number of blades and the number of elements circumferentially in the model. Then you will be prompted for the circumferential spacing. Enter a carriage return to use equal spacing. Enter circumferential percentages with the sum being less than 1.0. For example, for 4 circumferential elements, biasing percentages of 0.10 0.23 0.34 with place the nodal sections at 0, 10%, 33%, 67%, and 100% of the sector angle. The 3D disk model is then generated. Also the disk weight is calculated based on a density of 0.3 lb/in^3. The output is a UIF (partname.3uf) of the disk model. The model is a sector model with symmetry boundary conditions. Optionally a Patran Neutral File (partname.pnf) and/or a CSTEM input deck (partname.cst) can be written.

1.6.1.3.5. EXAMPLE

Following is an example of running the Disk Model Generator. See Figures 1.6.1.3.2 - 1.6.1.3.3 for plots of the disk model generated.

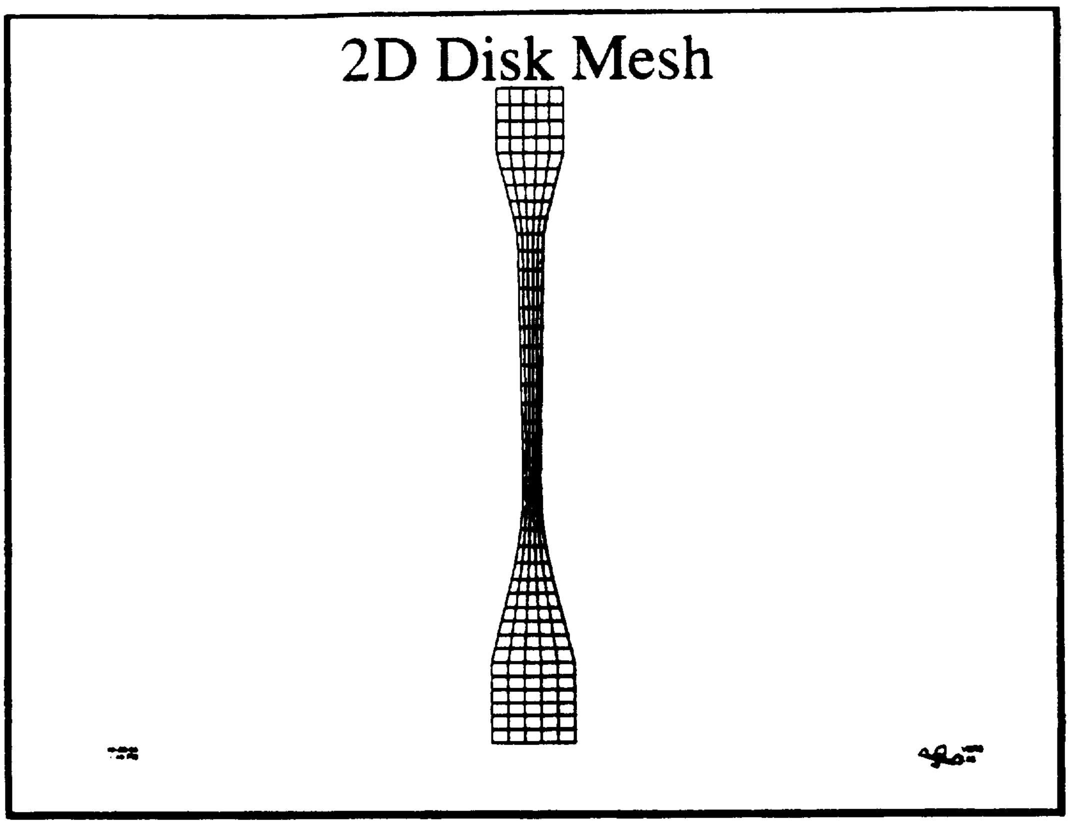


Figure 1.6.1.3.2

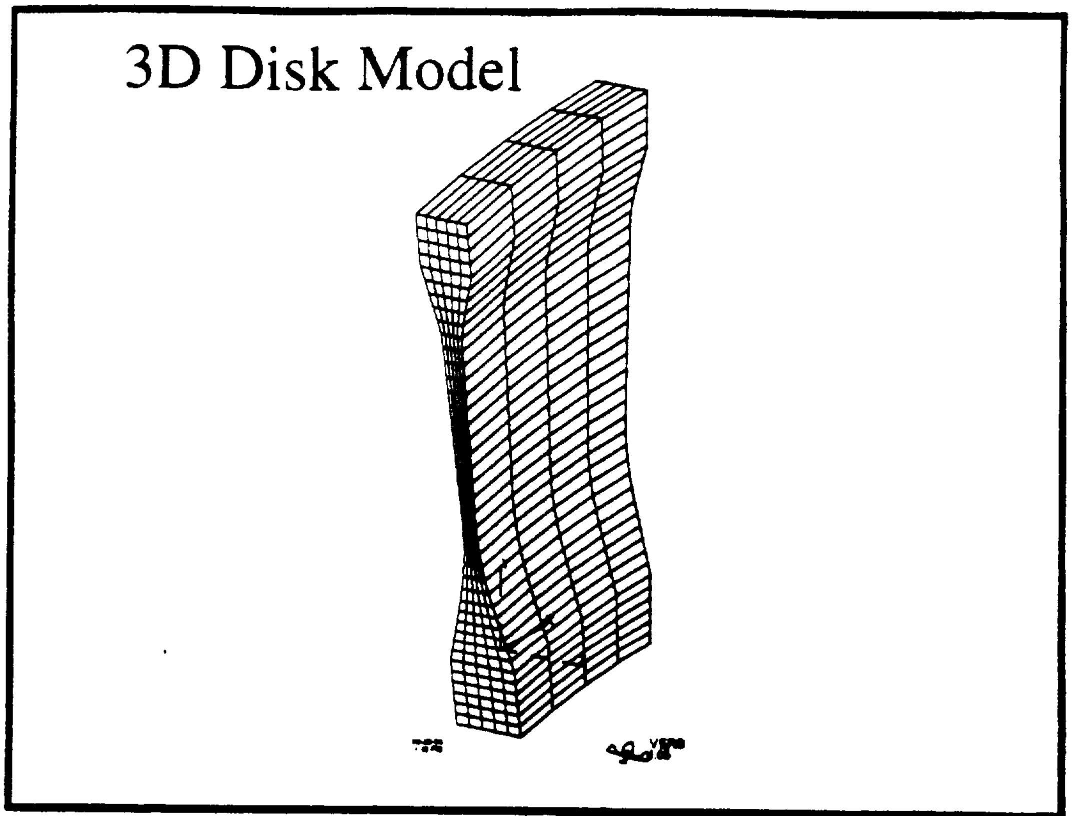


Figure 1.6.1.3.3

Disk Model Generator Example (Input is Underlined)

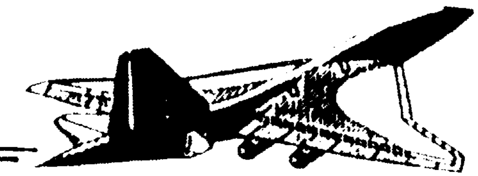
```
> xcosmo
    ###################################################################
                  CCCCC    OOOOOO    SSSSSS   MM    MM   OOOOOO
                  CC       OO  OO    SS       MMM MMM    OO  OO
                  CC       OO  OO    SSSSSS   MM M MM    OO  OO
                  CC       OO  OO        SS   MM M MM    OO  OO
                  CCCCCC   OOOOOO    SSSSSS   MM    MM   OOOOOO
    IT IS  1:51 PM ON 06-30-94 SYSTEM c0401          VERS I.01 06-30-94
    ###################################################################
     This is Version I.01 of COSMO (Production).
     Type NEWS for the latest COSMO News - last update 06-29-94.
          Type NEWS for more information.

                     AVAILABLE SUB-MENUS
    (?)  MENU, (Q)  QUIT, (SYSTEM) SYSTEM
     0 EXIT FROM COSMO

     1 COSMO GEOMETRY                       2 SIESTA FUNCTIONS
    31 CUSTOMIZE COSMO

         CURRENT MAXIMUM SUB-MENU IS 31

         YOU ARE IN THE MAIN MENU (0)
    ENTER DESIRED SUB-MENU BY NUMBER OR ?,G,Q,SYSTEM
DISK
    #############################################################
          COSMO DISK MODEL GENERATOR    VERS I.01  06-30-94
              ON c0401           AT  1:51 PM 06-30-94
    #############################################################

    ENTER THE PART NAME (UP TO 8 CHARACTERS)
part1
    THE CURRENT DISK MODEL PARAMETERS ARE:
```

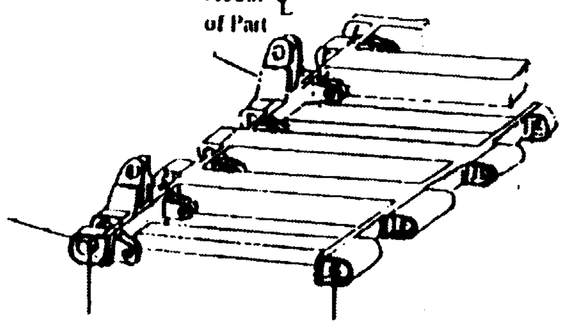
APPENDIX B

HSCT NOZZLE GEOMETRIES

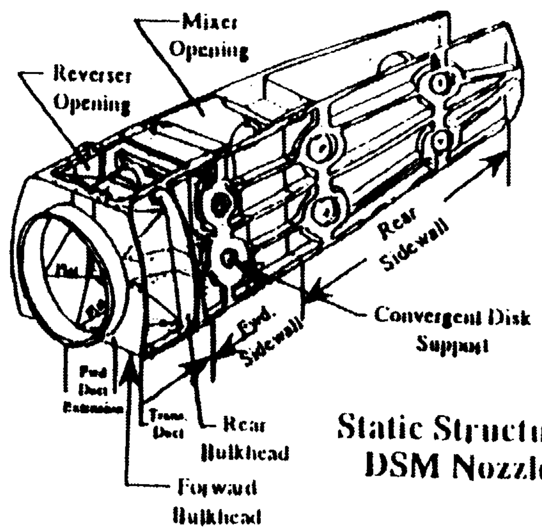
HSR-EPM

DSM Nozzle Components

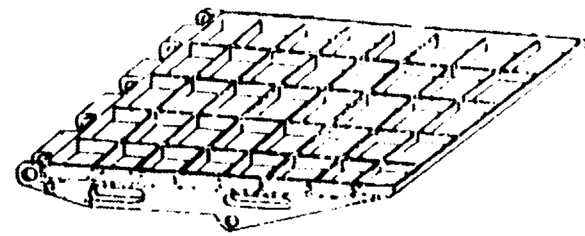




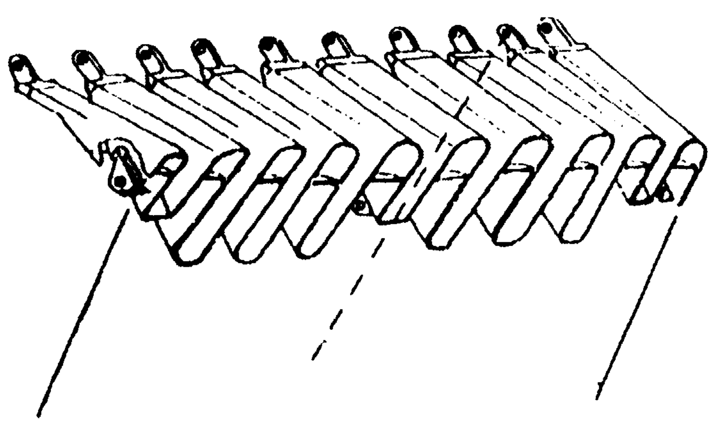
Forward Divergent Flap



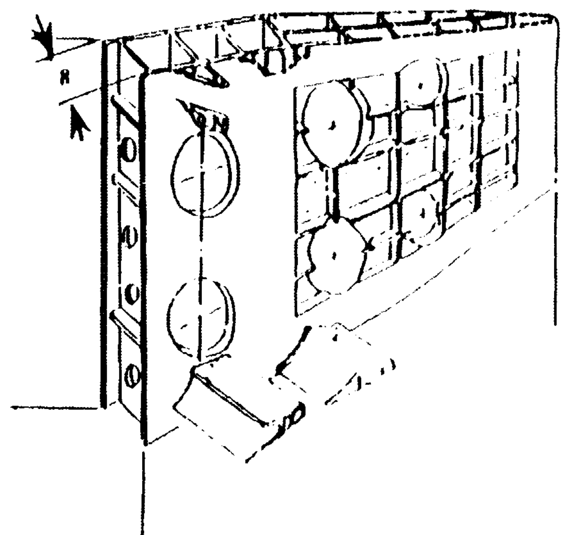
Static Structure
DSM Nozzle



Rear Divergent Flap



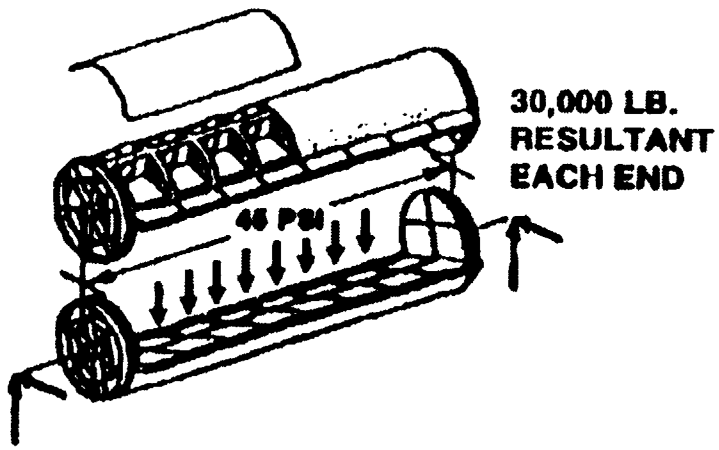
Lower Mixer



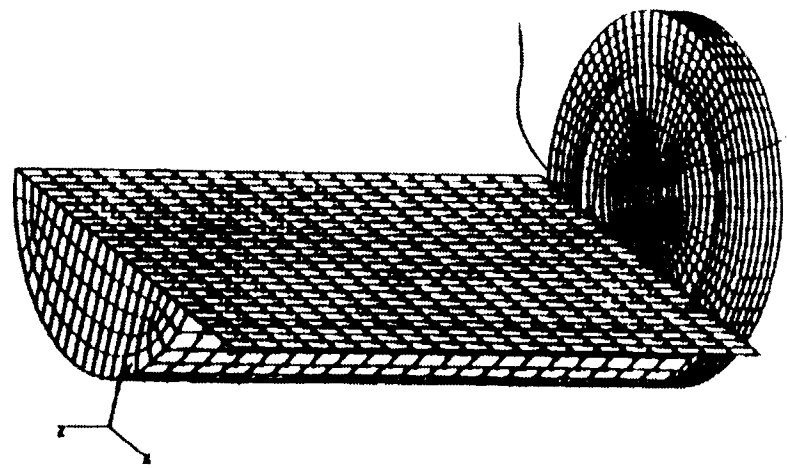
EPM

NOZZLE PROGRAM

STRUCTURAL MODEL OF DSM CONVERGENT FLAP DEFINED



DSM CONVERGENT FLAP DESIGN

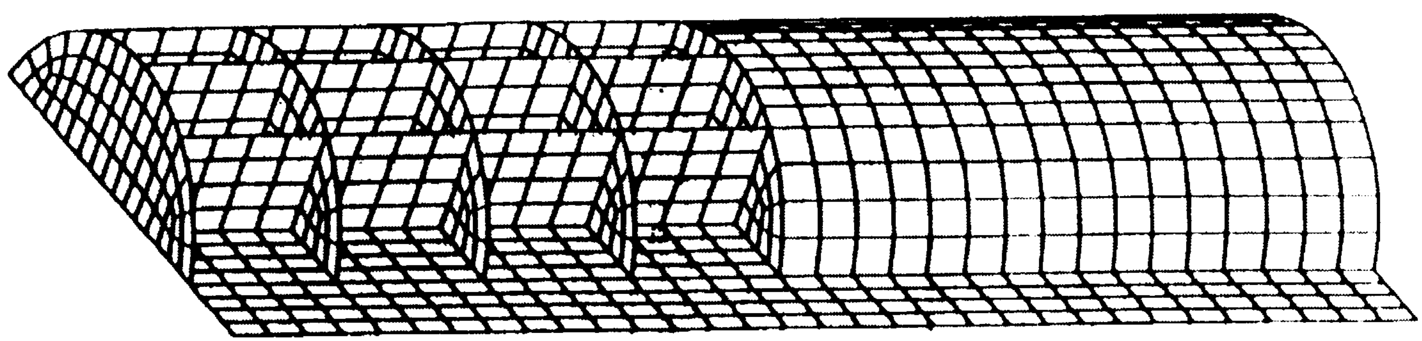


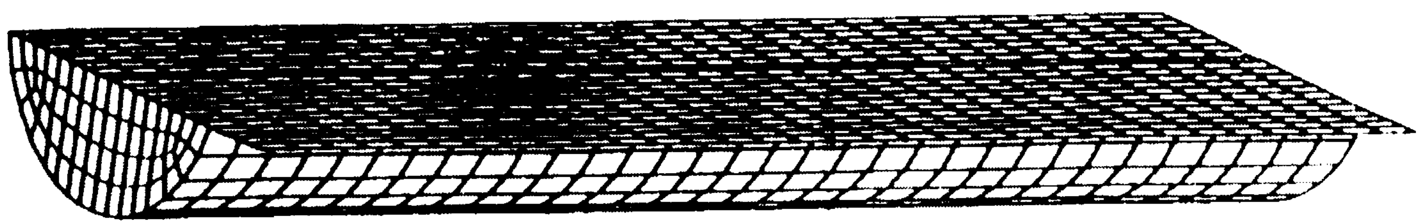
CONVERGENT FLAP STRUCTURAL MODEL

EPM

NASA ALTERNATIVE APPROACHES

2D CONVERGENT FLAP FINITE ELEMENT MODEL

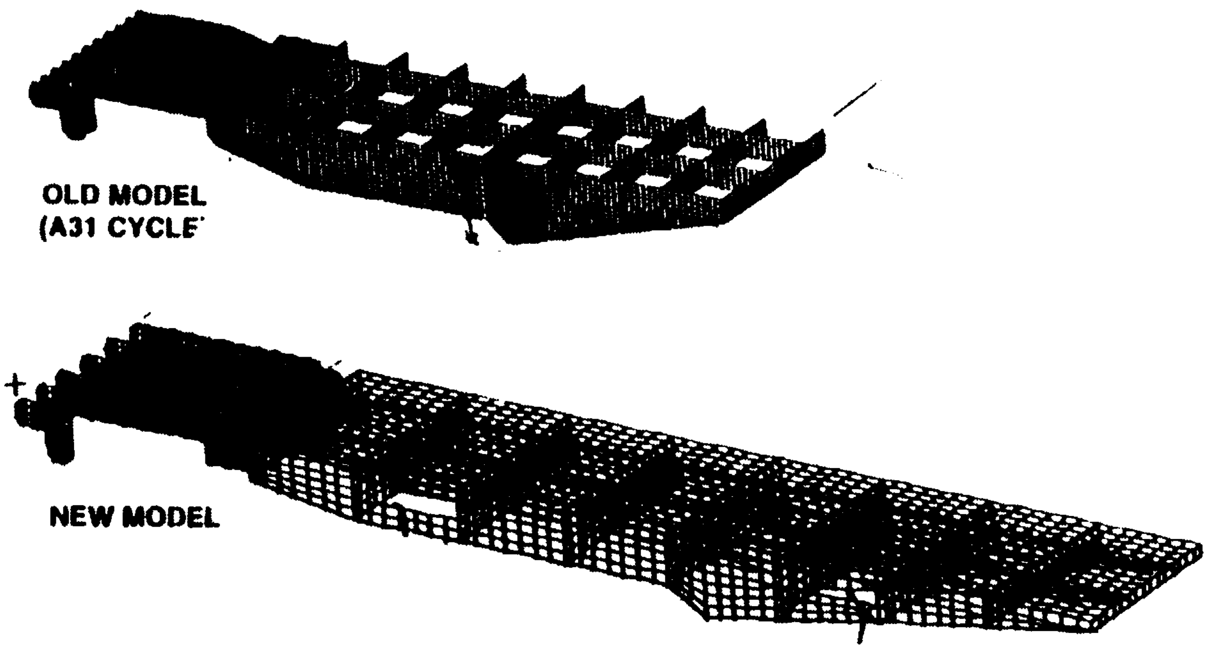




EPM

NOZZLE PROGRAM

STRUCTURAL MODELS FOR FORWARD AND AFT DIVERGENT FLAP DEFINED FOR DSM NOZZLE

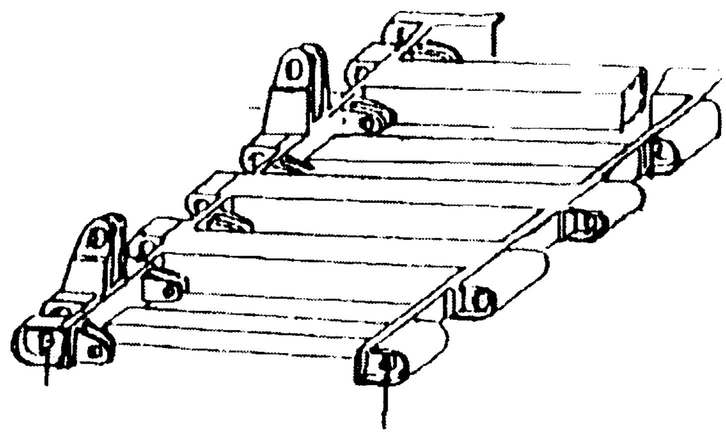


- LENGTH OF AFT DIVERGENT FLAP INCREASED FROM 99 TO 151 INCHES
- ESTIMATED WEIGHT INCREASE FOR AFT DIVERGENT FLAP IS 50 PERCENT

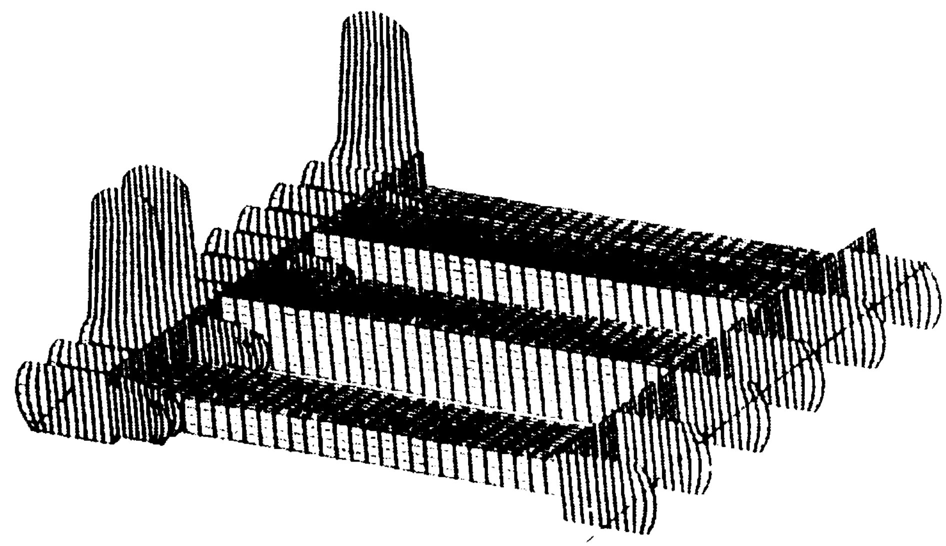
EPM

NOZZLE PROGRAM

STRUCTURAL ANALYSIS OF DSM DIVERGENT FLAP WILL DETERMINE MATERIALS/DESIGN TRADEOFFS



FORWARD DIVERGENT FLAP

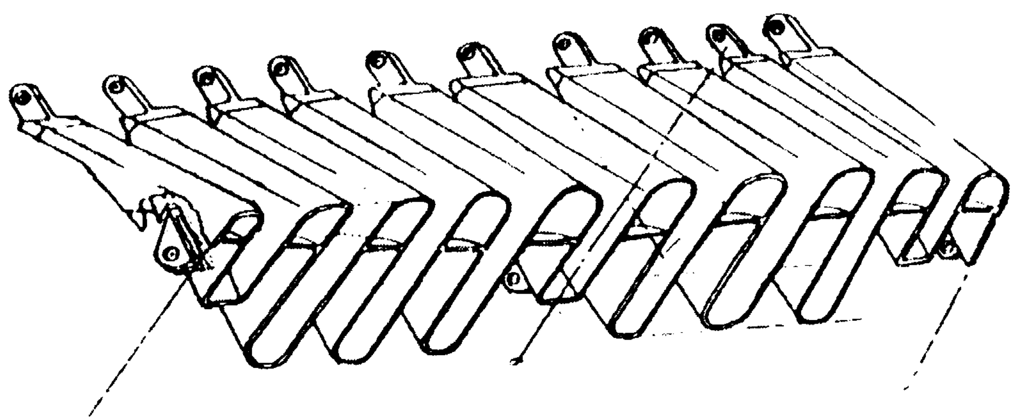


NASTRAN MODEL

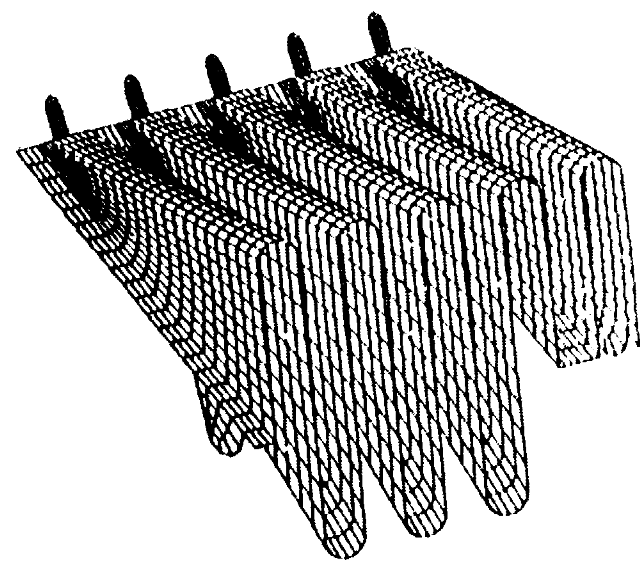
EPM

NOZZLE PROGRAM

STRUCTURAL MODEL OF DSM SHOOT/MIXER DEFINED



DSM SHOOT/MIXER DESIGN

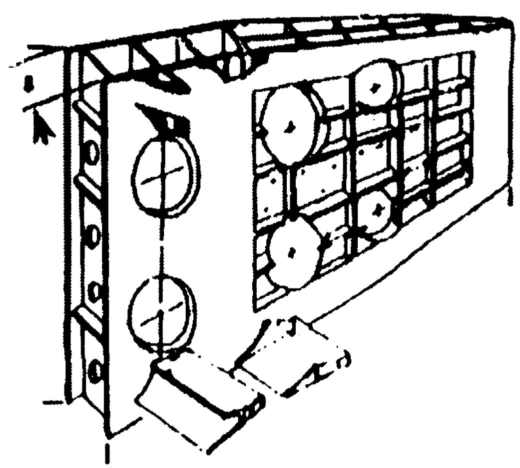


STRUCTURAL MODEL

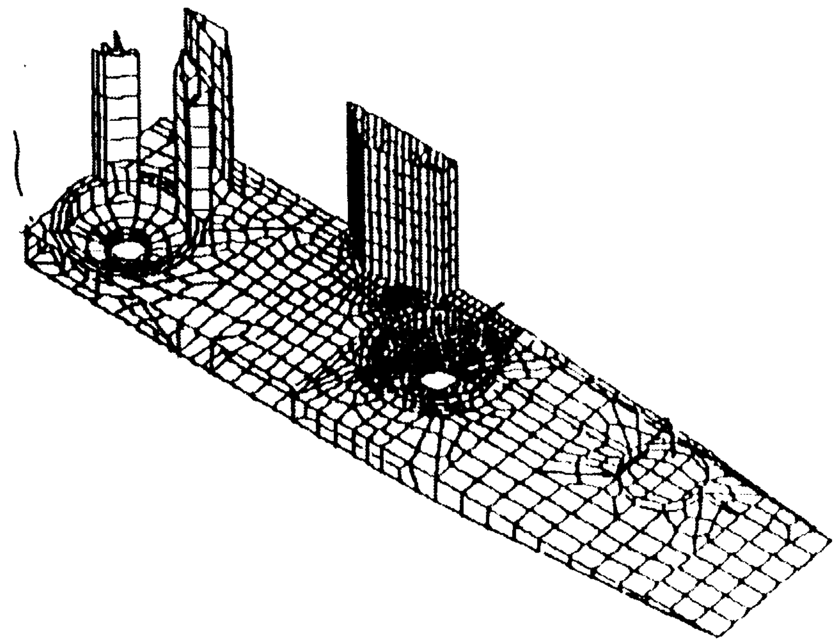
EPM

NOZZLE PROGRAM

STRUCTURAL MODEL OF DSM SIDEWALL DEFINED



DSM SIDEWALL DESIGN



SIDEWALL STRUCTURAL MODEL

APPENDIX C

EZFRAMES USER'S MANUAL

1. INTRODUCTION

The EZFRAMES program is a tool for building finite element models of circular engine frames or other similar structures. The program requires a minimum amount of input data to describe the frame geometry, material properties, boundary conditions, and analysis controls. The program generates a SIESTA Unified Input File (UIF) of the complete 3D finite element model and can continue by automatically spawning a sequence of finite element analyses of the model, postprocessing the results to obtain flexibility coefficients which characterize the frame stiffness. Input can be obtained from a keyword style input file or from an interactive session of questions and answers. In either case, a new keyword file is generated which can be used to reconstruct the model if desired.

The EZFRAMES program is an enhancement and extension of the original ZFRAMES program. The basic concepts utilized by the original authors of ZFRAMES remain unchanged. These include:

1) Definition of frame geometry by specification of ring, strut, and panel entities
2) Automatic generation of 3D finite element model comprised of beams, plates, and rigid connectors
3) Automatic creation of a SIESTA UIF of the finite element model
4) Capability to generate struts with axial and tangential lean and arbitrary positioning
5) Capability to vary individual strut properties
6) Beam section properties calculated for standard solid or hollow sections
7) Capability to accept general beam section properties
8) Capability to modify orientation of beam sections relative to beam plane definitions
9) Standard loads and displacement boundary conditions applied to the rings
10) Echo of input allowing automatic reconstruction of the model

The main enhancements or extensions from the original ZFRAMES program involve the following:

1) Capability to read inputs from a keyword style input file
2) Removal of inherent model size restrictions due to the node numbering scheme
3) Multiple strut sets
4) Multiple inner and outer connecting rings for struts
5) Multiple beam struts, allowing variation in twist and temperature
6) Multiple plate sections attached to a ring
7) Capability for linear variation of thickness and temperature across plate sections
8) Capability to define symmetrically spaced cutouts in rings and plate sections
9) Capability to include external UIF data
(for generic material property specification, etc.)
10) Automatic generation of flexibility coefficients of the frame

2. INPUT

The primary and recommended method of input to EZFRAMES is through the use of a keyword driven input file. The use of a keyword driven file allows for easy modification and relieves the user from the need to follow a strict input order. The keyword file is read in its entirety before EZFRAMES begins a check of the stored data for completeness and consistency, then continues with the frame modelling and analysis. For this reason the order in which various entities are input and defined is not restricted in any way.

However, there are some input rules which must still be followed for keyword input files. Keyword input files for EZFRAMES are similar in many ways to SIESTA keyword style UIF files. They are characterized by one or several 4 character keywords which occur on a line of input, followed by one or several lines of data that correspond to the keywords. The keywords define what the data is and where it occurs relative to other data items on the line.

There are two types of keywords recognized by EZFRAMES: primary and secondary. Primary keywords define the entity or scope of subsequent secondary keywords. A primary keyword must be the first keyword on a line. There are no values associated with primary keywords. They simply set the stage for the secondary keywords that follow and remain the active primary keyword until another primary keyword is read.

Secondary keywords may occur on the same line as a primary keyword. More than one secondary keyword may occur on a single input line. There is typically a data value that is expected to be read corresponding to a secondary keyword. An exception to this is the BASE keyword, which has no associated value. The BASE keyword indicates that subsequent values corresponding to secondary keywords appearing on the same line as the BASE keyword are to be used as default values for all entities of the active primary keyword. The BASE keyword can be thought of as playing the role of the NAME keyword, except that there is no specific NAME value and BASE is not active beyond the keyword line in which it appears (i.e. it must be repeated on each line if multiple keyword lines are use to define all BASE values).

With this basic understanding of the workings of the EZFRAMES keyword file, it is probably best to refer to the EZFRAMES keyword summary listed in Section 10. The keyword summary also lists existing default values for quantities for which defaults are available. If values with available defaults are not specified, the defaults will be used. If quantities with no defaults are not specified, EZFRAMES will stop and specify that an error has resulted.

EZFRAMES can also be used in an interactive mode, in which the program asks for input item by item. Regardless of the input mode, a keyword input file is written to the file Zkey, so the interactive mode can be used to generate a keyword file. If the interactive mode is used, an echo of the interactive responses is written to the file Zf10. This file may be used as input to reconstruct the EZFRAMES model from the interactive mode, although use of the keyword input mode is preferred.

A sketch of an example bypass frame to be modelled is shown in Figure 2. Figures 3. and 4. show the resulting EZFRAMES model for the keyword input file listed in Figure 5. The keyword input file illustrates the use of the BASE keyword, multi–ring struts, inclusion of an additional UIF fragment to define material properties, and non–sequential numbering of entities.

3. GEOMETRY DEFINITION

Geometry definition in EZFRAMES is based on the specification of the axial and radial location of rings, which are modelled as curved beam segments. Struts and panels are defined relative to the rings they connect. The number of beams in a ring (divisions) is determined by the number of struts connected to the ring and the number of plates to be placed between the struts. Ring divisions are determined by starting at a ring and tracing along the panels connected to that ring to the most forward (end 1) ring in the series of panels. The ring with the maximum number of struts in that series of panels is used to set the number of strut divisions for every ring in the series. All the rings in the model are checked in input sequence to ensure the number of divisions is defined for every ring. Any ring without a connecting panel should have a connecting strut or vice versa. For models without struts, the number of ring divisions is calculated based on the plate aspect ratio.

Strut sets are defined by specifying the rings connected to the inner and outer strut ends. More than one ring may connect to the same strut set, and more than one strut set may connect to the same ring. A strut set is located by axial and radial offsets from the <u>first</u> connecting ring at each end. Struts may be comprised of multiple beams, primarily to enable changing twist angles along the span of the strut. Axial lean of the strut is specified by the axial offset at each end of the strut. Strut positioning is specified by the angles at which the first strut in a set connects to the inner and outer rings. Tangential lean can be set by either specifying the first strut position at both the inner and outer rings or by specifying a tilt angle along with only the inner or outer strut position. The inner and outer rings are rotated by their respective angles to accommodate the strut positioning. If more than one strut set connects to a ring, the ring is rotated according to the first strut set to which the ring is connected at the inner strut end. Base values for number of struts, offsets, number of beams per strut, start angles, and number of divisions between struts can be set using keyword input. These base values are used for every strut set created unless specifically changed for an individual strut set.

Panels are defined by specifying a ring connected at each end. The panels are located by axial and radial offsets from the connecting ring at each end. Any number of panels may connect to a particular ring. The number of plate rows between the panel ends is set by specifying the maximum aspect ratio for plates within the panel. The aspect ratio along with the ring divisions determines the allowable plate length. A minimum of two plate rows will be used for panels between two rings connected to the same strut set regardless of the aspect ratio calculation. Base values for offsets and aspect ratio can be set using keyword input. These base values are used for every panel created unless specifically changed for an individual panel.

Circumferentially spaced cutouts or holes can be specified for rings and panels. The cutouts are formed by first assuming a complete ring or panel is to be created, then skipping the formation of elements at the cutout location. The cutout or hole is specified by a packed hole code indicating the existence or non–existence of individual elements in a sector of the complete ring or panel. The number of sectors must be a multiple of the number of ring divisions. The packed code is a sequence of 0's and 1's where 0 indicates a hole (i.e. no element) and 1 indicates an element. The number of 0's and 1's in the packed code is the number of ring divisions divided by the number of sectors. The first element of a sector corresponds to the left–most digit in the packed code and the last element of a sector corresponds to the right–most digit in the packed code. Any missing digits are filled as 0 on the left side of the packed code, so that formation of the first elements in a sector will be skipped if there are missing digits. Specifying a hole code as 0 will completely eliminate a ring. The packed code is stored as an integer so there is a limit to the number of digits which can be stored in this way. No more than 10 digits should be used in the packed hole code.

4. COORDINATE SYSTEM

The coordinate system used by EZFRAMES is a right-handed rectangular X,Y,Z system. The Y axis is vertical and the Z axis is axial. When preparing a model for EZFRAMES it is easiest to sketch a cross section in the Y–Z plane (X into the paper) so that the radial direction is vertical and the axial direction is horizontal. For consistency with typically used rotordynamics programs, the Z axis should be pointing aft. The sketch can be used to axially and radially locate the rings and the axial and radial offsets of struts and panels.

It is important to keep this coordinate system in mind when defining the loads since direction is important. Specification of displacement components is done with respect to an axial, radial, tangential cylindrical coordinate system that is obtained by transforming the rectangular X, Y, Z system. This axial, radial, tangential coordinate system corresponds to the SIESTA cylindrical coordinate system, with X' axial, Y' radial, and Z' tangential. This transformation is done internally and is only apparent to the user when the X, Y, or Z keywords are used to specify displacement loadings. In such a case the X, Y, Z keywords refer to the transformed coordinate system, so it would be more straightforward to use the AX, RAD, TANG keywords instead of X, Y, Z.

5. NODE AND ELEMENT NUMBERING

Node and element numbers are assigned first to struts, then rings, then panels, then rigid connectors. No specific node or element number blocks are reserved for particular entities, although convenient starting values are determined for ring, panel, and reference nodes based on the model size. Element numbers are related to the node numbers contained in the element.

Strut node numbers begin with 1 and are created in sequence tangentially counterclockwise as viewed from the coordinate system origin looking in the positive axial direction, then progressing from inner to outer for each strut set. The first inner and outer node of a strut set are at the specified starting inner and outer angular position, respectively, for the strut set. Strut beam element numbers are the same as the strut inner node number.

Ring node numbers begin at a convenient hundreds value and are numbered sequentially in the tangential counterclockwise direction, progressing from ring to ring in input order. The location of the starting node number of a ring corresponds to the starting angle for the first strut set using the ring as an inner connecting ring. Ring beam element numbers are the same as the first (i.e. lower) node number in the element.

Panel node numbers begin at a convenient thousands value and are numbered sequentially in the tangential counterclockwise direction, progressing plate row to plate row from the first connecting ring to the second connecting ring. Node numbers are defined for panels in input sequence. Plate element numbers are the same as the first (i.e. I joint) node number in the element.

Reference nodes are created for the strut and ring beam elements in a sequence similar to the structural nodes. If possible, reference nodes begin at 8001 if the model is not so large as to require this value for structural nodes. Otherwise, a convenient thousands value after the last plate node is used.

Rigid connector numbers begin with the same value as the reference nodes. The rigid connectors between struts and rings are created first, progressing tangentially about the inner strut ends, then outer strut ends, for each strut set in input order. Rigid connectors for the panels are created next, progressing tangentially for the first end, then the second end, for each panel in input order.

6. ELEMENT PROPERTIES

Element properties consist of beam moments of inertia and torsional constants, beam orientation definitions, thicknesses of plates, material types, and temperatures. Base values can be assigned using keyword input which will be used for every element of a certain entity type unless specifically changed for an individual entity.

Definition of maximum and minimum moments of inertia and torsional constants are required for the beam elements of struts and rings. These quantities are used to define unique beam types, then the beam types are assigned to specific rings and strut sets. The moments of inertia and torsional constant can be internally calculated for certain standard beam cross sections by defining the cross section dimensions. This option is available for solid or hollow rectangular and elliptical cross sections, where the necessary cross section dimensions are the major and minor axis lengths and the wall thickness or –1 for solid sections.

Beam elements are oriented by defining the BETA angle, which specifies the angle of the min moment of inertia axis (i.e. major cross section axis) with respect to the plane of the beam. The plane of the beam is defined by EZFRAMES using reference nodes created specifically for that purpose. For zero BETA angle, the min moment of inertia axis lies in the plane of the beam. Positive BETA angle corresponds to a right–handed rotation about the I–J beam axis using the I joint as the origin. The SIESTA manual contains a figure which illustrates the definition of the angle BETA. Figure 1. shows positive BETA angle orientations for strut and ring beam elements.

For struts, the plane of the unrotated beam lies in the radial–axial plane and the BETA angle defines the twist of the strut. Strut beams are defined from inner to outer (I to J) with the O joint offset in the +Z (axial) direction. Positive BETA angle will rotate the end of the strut beam cross section nearest to the coordinate system origin to the right as viewed from the origin in the positive axial direction. For multiple beam struts the value of BETA can change linearly from the inner end to the outer end, with each beam assigned the BETA value at the beam midspan.

For rings, the plane of the unrotated beam lies in the radial–tangential plane and the BETA angle defines the angle of the beam from radial. Ring beams are defined in the counterclockwise direction as viewed in the positive axial (Z) direction. Positive BETA angle will rotate the radially inner end of the ring beam cross section in the negative axial direction.

Plate thicknesses can vary linearly along a panel from end #1 to end #2. Temperatures can vary linearly for plates and struts, but are constant for rings. The temperature of rigid connectors corresponds to the temperature of the ring to which they connect. The temperature of rigid connectors for struts with multiple connecting rings is the average of all the rings connected to the strut end.

Each entity is assigned a material number which then relates a set of material properties to that entity. General material properties for a particular material number are defined using the SIESTA UIF convention and included with the EZFRAMES created UIF. This can be done automatically by specifying the filename of the partial UIF to be included. ESPEC properties can be accessed in this same way. ESPEC properties can also be defined without including a partial UIF by specifying a negative material code that corresponds to an ESPEC table included in EZFRAMES. This ESPEC table can be listed from the interactive mode of EZFRAMES. Rigid connectors are assigned the material of the ring to which they connect. For struts with multiple connecting rings, the material of the first connecting ring is used for the rigid connector.

7. BOUNDARY CONDITIONS

Boundary conditions can be applied to rings only. Deflection and standard load boundary conditions can be applied. They can be applied to more than one ring. Load conditions can be applied to the same ring if they result in forces in different directions. Two load conditions with forces in the same direction applied to the same ring will not add.

The most common deflection boundary condition is to completely fix a ring. This can easily be done with the LOAD primary keyword active by using the FIXR keyword and specifying the ring number to be fixed. Various combinations of deflection components can be fixed or displaced some non–zero value using the DEFL primary keyword. Deflection components are assumed to be in an axial, radial, tangential cylindrical coordinate system, which requires a transformation from the usual X,Y,Z rectangular coordinate system of EZFRAMES. This cylindrical coordinate system is defined so that X' is axial, Y' is radial, and Z' is tangential. In such a case the X, Y, and Z keywords refer to the cylindrical coordinate system, so it is more straightforward to use the AX, RAD, TANG keywords instead of X, Y, Z to define deflections.

Standard load boundary conditions can be requested for which the specific nodal force values are internally calculated and applied. For all except the uniform axial load, a phase angle can be applied which rotates the location of the maximums and minimums from the top, bottom, side locations to any desired angle. The standard loading conditions are:

1) Uniform axial load
   This loading simply applies a constant force to each node on the ring in the axial direction. The user inputs the total force desired and the program divides this among the nodes on the ring.

2) Sinusoidal axial load
   This applies a sinusoidally varying load in the axial direction. The load is minimum at the top and increases sinusoidally to a maximum at the bottom for a phase angle of zero. The user inputs the total force desired and the ratio of the minimum peak at the top to the maximum peak at the bottom.

3) Sinusoidal transverse load
   This choice applies a load in the vertical direction which is maximum at the top and bottom and zero at the sides for a phase angle of zero. The user inputs the total force.

4) Sinusoidal circumferential shear load
   This load is similar to the sinusoidal transverse load, but the loads are applied to each node along the tangent to the ring at that point. The user inputs the total vertical force desired. The program automatically divides up the vertical force among the ring nodes and calculates the appropriate horizontal force to achieve a tangential resultant.

5) Sinusoidal overturning moment
   This selection applies loads which vary sinusoidally and combine to create an overturning moment about the X axis. The loads are maximum at the top and bottom and are zero at the sides for a phase angle of zero. The user inputs the total overturning moment desired. A positive input results in forces in the positive axial direction on the top of the frame and forces in the negative axial direction on the bottom of the frame. The resulting positive moment is counterclockwise, looking in the positive X direction.

8. ANALYSIS

The translation capabilities of SIESTA allow the generation of input to several analysis programs once the UIF of the frame model is created. The user can always use the generated UIF for this purpose and run any specific analysis desired. There are also procedures imbedded within EZFRAMES that will automatically perform a standard set of analysis cases and postprocess the results to generate flexibility coefficients for use in a system dynamics analysis of an engine with the modelled frame.

There is currently only one type of standard analysis available. The type of standard analysis is indicated by the value assigned with the CASE keyword under the LOAD primary keyword. The CASE keyword will cause multiple load cases of the type needed to generate the desired flexibility coefficients to be written to the UIF.

The POST primary keyword is used to specify an inner and outer ring of nodes used to postprocess the analysis results for the flexibility coefficients. Each ring of nodes generally corresponds to the location of one end of a linear spring. Nodes of a ring or within a panel can be specified. If nodes in a panel are desired an axial location must also be specified, and the row of nodes in the panel nearest to that axial location will be used for postprocessing. Multiple postprocessing locations can be requested to generate the multiple flexibility coefficients for each linear spring desired.

It is possible to use one of several codes to perform a standard analysis. The user can choose between ANSYS, NASTRAN, and MASS by using the CODE keyword under the CTRL primary keyword. ANSYS is the default code used if left unspecified.

9. OUTPUT

The primary output of EZFRAMES is the SIESTA UIF. The UIF is stored in file 31 (f31.dat) and if the program completes normally, this file will be renamed to *Zframes.uif.*

The generated keyword input file is written to the file *Zkey.* If constructed using the interactive mode, a record of the interactive responses is written to the file *Zf10.*

There are several warnings that can possibly be generated when EZFRAMES checks the stored data for completeness and consistency. Any time a default value is used, a warning is generated. Unspecified values with no default will cause an error message to be written. These warning and error messages will be written to the file *Zwarns* as well as to the screen.

If the standard analysis is performed, the results at the requested inner and outer node rings and other pertinent postprocessing results will be written with the flexibility coefficients to the file *Zflex.*

10. KEYWORD SUMMARY

RING	Used to get RING data	
NAME	RING number	
AX	Axial coordinate of RING	default = 0
RAD	Radial coordinate of RING	default = 0
HOLE	Packed hole code for a sector, 0=hole, 1=element	
NSEC	Number of sectors in ring for hole calculations	
PHI	RING starting angle (overrides strut ANGL)	
BASE	Used to set defaults for all undefined RINGS	
BTYP	BEAM type of RING	default = 1
BETA	X–sectional angle to principal axis (constant)	default = 0
TEMP	STRUT temp (constant)	
MATL	Material code (<0 to correspond to internal ESPEC table)	

STRUT	Used to get STRUT data	
NAME	STRUT set number	
IDST*	Individual STRUT number within a set	
RNGI	Inner RING number (multiple allowed)	
RNGO	Outer RING number (multiple allowed)	
BASE	Used to set defaults for all undefined STRUTs	
NUMS	Number of STRUTs in set	
BTYP*	BEAM type for STRUT	default = 2
NSUB	Number of BEAMs per STRUT	default = 1
MATL*	Material code (<0 to correspond to internal ESPEC table)	
BETA	X–sectional angle to principal axis (constant)	default = 0
BETI*	angle at inner RING (for multi–BEAM STRUTs)	
BETO*	angle at outer RING (for multi–BEAM STRUTs)	
TEMP	STRUT temp (constant)	
TMPI*	STRUT temp at inner RING end	
TMPO*	STRUT temp at outer RING end	
TILT	Circumferential tilt of strut	

When used for named STRUT sets the following keys refer to inner or outer based on last read: RNGI or RNGO

ZOFF	Axial offset from from 1st RING	default = 0
ROFF	Radial offset from from 1st RING	default = 0
NDIV	Number of divisions on RING between STRUTs	default = 2
ANGL	Circumferential angle of 1st STRUT on RING (0=TDC)	default = 0

(* These keywords also apply to individual strut definitions, IDST)

PANS	Used to get PANel (casing) data	
NAME	Name of PANel	
RNG1	First (forward) RING	
RNG2	Second (aft) RING	
HOLE	Packed hole code for a sector, 0=hole, 1=element	
NSEC	Number of sectors in PANel for hole calculations	
BASE	Used to set defaults for all undefined PANels	
RINC	Default increment from RING 1 to RING 2	
THIK	PANel thickness (constant)	
THK1	PANel thickness at RING 1 end	
THK2	PANel thickness at RING 2 end	
ASPR	Max aspect ratio of plates in this PANel	default = 3
MATL	Material code (<0 to correspond to internal ESPEC table)	
TEMP	PANel temp (constant)	
TMP1	PANel temp at RING 1 end	
TMP2	PANel temp at RING 2 end	

When used for named PANels the following keys refer to RING 1 or RING 2 based on last read: RNG1 or RNG2

ZOFF	Axial offset between PANel and RING beams	default = 0
ROFF	Radial offset between PANel and RING beams	default = 0

LOAD	Used to load RINGs
RNGN	RING number to load
FIXR	Constrained RING (value is RING to be fixed)
ZUNF	Uniform axial total load value
ZSIN	Sinusoidal axial total load value
VSIN	Sinusoidal transverse shear total load value
CSIN	Sinusoidal circumferential total load value
MOVR	Overturning moment total load value
RATO	Top to bottom ratio of sinusoidal axial load
CASE	Automatic load case setup flag
ANGL	Load phase angle for last read load (default=0., TDC)
DEFL	Used to specify RING deflections
RNGN	RING number with specified deflections
X	Tangential deflection (same as TANG)
Y	Radial deflection (same as RAD)
Z	Axial deflection (same as AX)
ROTX	Rotation about X [degrees]
ROTY	Rotation about Y [degrees]
ROTZ	Rotation about Z [degrees]
TANG	Tangential deflection (same as X)
RAD	Radial deflection (same as Y)
AX	Axial deflection (same as Z)

BEAM	Used to get BEAM properties for RINGs and STRUTs	
NAME	BEAM type name	
AREA	X–sectional area	
IMIN	X–sectional Imin	
IMAX	X–sectional Imax	
KTOR	X–sectional torsional constant	

These parameters will cause calculation of AREA, IMIN, IMAX, KTOR

STYP	Cross–section type (1=rectangular, 2=elliptical)	default = 1
THIK	Wall thickness for hollow sections, –1 for solid	default = –1
LMAX	Major axis length (valid for STYP= 1 or 2)	
LMIN	Minor axis length (valid for STYP= 1 or 2)	

CTRL	Used to get analysis control information	
TREF	Reference temperature	default = 70
TCON	Constant model temperature	
CODE	Analysis code (1=ANSYS, 2=NASTRAN, 3=MASS)	

POST	Used to get post–processing locations
ROTR	RING number to use as outer circumference location
RINR	RING number to use as inner circumference location
POTR	PANel number to use as outer circumference location
PINR	PANel number to use as inner circumference location
AXOT	Used with POTR to determine nodal row for outer circumference
AXIN	Used with PINR to determine nodal row for inner circumference

---

UIF	Line following is a partial UIF to be included in the generated UIF
TITL	Line following is a title

===

Notes:

1) Primary keywords must be the first keyword on a line
(RING, STRU, PANS, LOAD, DEFL, BEAM, CTRL, POST, UIF, TITL)
2) Secondary keywords may appear on the same line as their primary.
3) BASE keyword must appear on the same line as base keywords being defined.
4) BASE keyword has no associated value.
5) BASE values will not overwrite any previously defined values.
(This means a BASE value can be set only once per primary type)
6) Specific definitions will overwrite any previously input value.
7) If no NAME is present on a keyword line, the last name input is used.
8) If no NAME is ever given for a primary type, NAME=1 is assumed.

9) Specific BEAM section properties on the same line as section parameters take precedence (i.e. overwrite the associated calculated property).

10) BEAM section parameters LMAX, LMIN must be provided and on the same line as STYP and THIK.

11) Existing defaults for parameters are shown immediately following the keyword definition. TREF is used as the default for temperatures. If these parameters are left unspecified, the default value will be used. If parameters without defaults are left unspecified, an error will result.

12) Rotation of nodes to radial–tangential is done only if some, but not all, deflection components are specified. Axial only will also not rotate nodes.

13) If TILT is used to specify STRUT circumferential tilt angle, either the inner or outer ANGL should also be given to properly locate the strut. Otherwise, the inner ANGL is defaulted to 0 and TILT applied from there. (All angles are in degrees.)

14) Material codes MATL entered as negative will not have properties defined by ESPEC. This is intended for use with the UIF keyword to append material properties in alternate UIF forms.

15) HOLE code for RINGs and plate PANelS is a packed code with a 0 or 1 for each element in a sector. The positions in the packed code correspond to locations in the sector progressing in the positive rotational direction from left to right in the packed code. NSEC for would generally be the same as the number of struts, NS, for strutted rings and panels connected to strutted rings. This is not assumed by EZFRAMES, however, so NSECP must be specified in order for hole calculations to be performed.

16) The angle keyword, PHI, for rings overrides the starting angular location inferred from struts attached to a ring. The starting strut remains attached at the angular location specified by ANGL, but the ring nodes will be numbered from the angle given in PHI. This should not be an often used keyword. Consideration should be given to the order of strut definition to set ring starting angle before using PHI. The first strut set using a ring as an inner connecting ring will define the ring starting angle. If the ring is not used as an inner ring, then the first strut set using the ring as an outer connecting ring will define the ring starting angle. First is based on the order of input, not NAME.

17) Specification of analysis code (CODE) can be done using either the numerical value or 1st 4 characters of the code name.

18) <u>Current limits:</u>

Max RINGs	= 100
Max STRUT Sets	= (Max RINGs)/2
Max PANelS	= (Max RINGs)–1
Max BEAM Types	= 2*(Max RINGs)
Max Unique Struts	= (Max STRUTs)*11
Max Strut End to Ring Connections	= 5

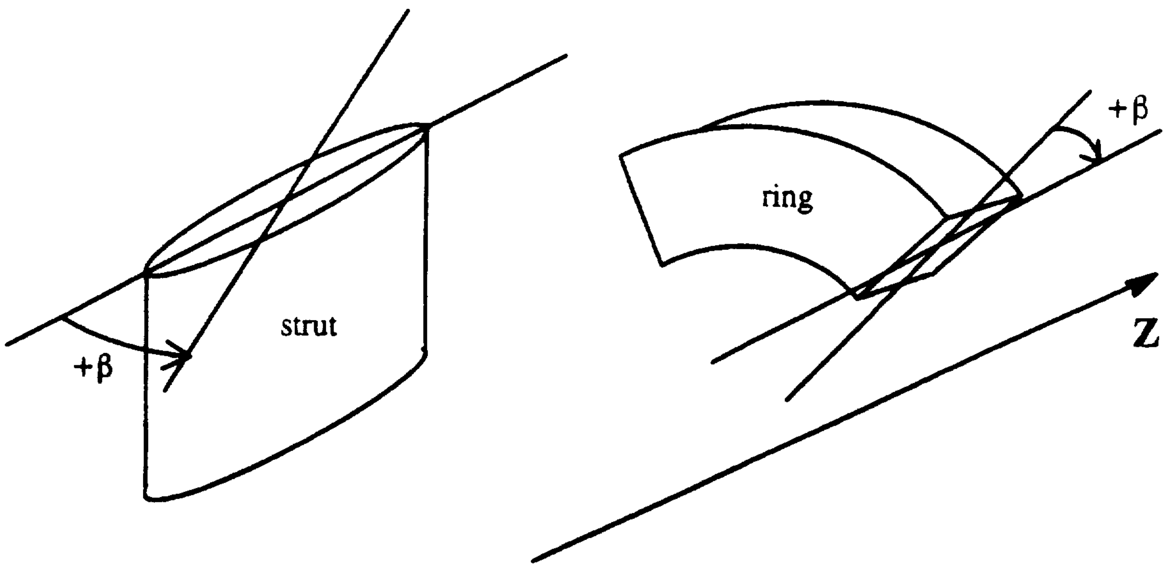


FIGURE 1. POSITIVE BETA ANGLE FOR STRUT AND RING BEAMS

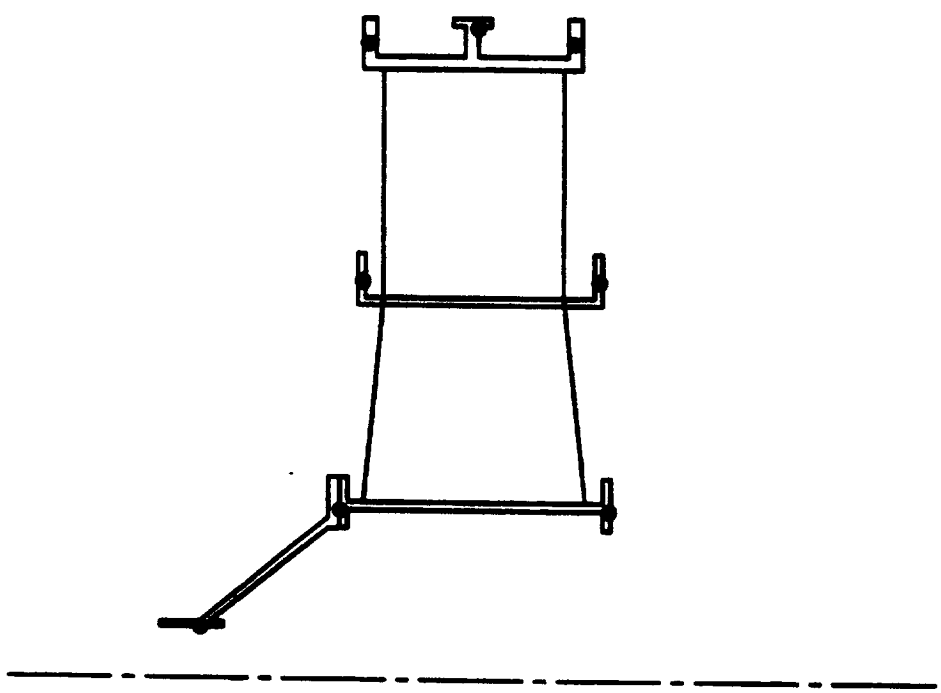


FIGURE 2. EXAMPLE BYPASS FRAME
● indicates ring location

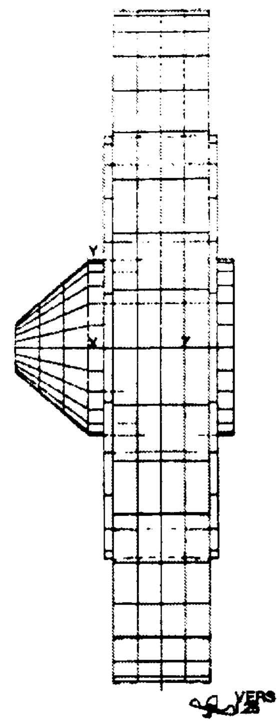


FIGURE 3. EZFRAMES EXAMPLE BYPASS FRAME MODEL

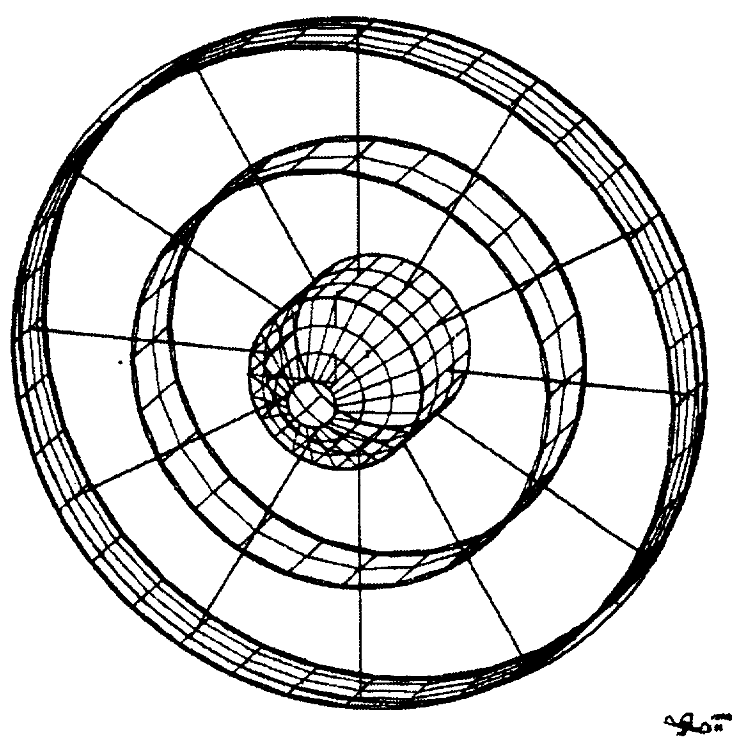


FIGURE 4. EZFRAMES EXAMPLE BYPASS FRAME MODEL

```
TITLE
   EXAMPLE BYPASS FRAME MODEL
RING             $ There are  8 RINGs
 BASE  TEMP MATL
    100.  1
 NAME AX RAD BTYP BETA
   1   0.0  11.0 5   0
   2 18.0  11.0 1   0
   3   2.0  26.5 2   0
   4 16.0  26.5 2   0
   5   3.0  42.0 3   0
   6 15.0  42.0 3   0
   7   9.0  43.0 4  90
 NAME AX RAD BTYP BETA TEMP
  10 -9.0  3.0 6  90  200.0
STRUT             $ There are  2 STRUT sets
 NAME NUMS BTYP NSUB MATL  &
  RNGI ZOFF ROFF ANGL NDIV BETI TMPI  &
  RNGO ZOFF ROFF ANGL NDIV BETO TMPO
  1  12  11   1  2 &
   1  9.0      .05     0  2  0.0    100.0    &
   3  7.0     -.35     0  2  0.0    100.0
  2  12  12   3  2 &
   3  7.0     -.35     0  2   0.0    125.0    &
   5  6.0     -.25     0  3  30.0    125.0
 NAME  RNGI  RNGO
  1    2    4
 NAME  RNGI  RNGO  RNGO
  2    4    7    6
PANS              $ There are  5 plate sections
  BASE  ASPR  MATL  TEMP
     3.0   1   100.
  NAME RNG1 ZOFF ROFF THK1 &
     RNG2 ZOFF ROFF THK2
 1 1  .0      .0     .1  &
   2  .0      .0     .1
 2 3  .0     -.25    .05  &
   4  .0     -.25    .05
 3 5  .0     -.25    .10 &
   7  .0    -1.25    .10
 4 7  .0    -1.25    .10 &
   6  .0     -.25    .10
 NAME RNG1 ZOFF ROFF THK1 TMP1 &
    RNG2 ZOFF ROFF THK2
10 10  .0      .0     .1     200.00    1  .0     -.1     .1
BEAM              $ There are 8 beam types
 NAME STYP THIK LMAX LMIN
  1   1  -1  0.5  0.25
  2   1  -1  0.75 0.1
  3   1  -1  0.5  0.1
  4   1  -1  1.0  0.1
  5   1  -1  0.5  0.5
  6   1  -1  1.0  0.075
 11   2  .1  14.0  1.0
 12   2  .1  12.0  1.0
CTRL  TREF
 70.000
UIF
matl.uif
```

<u>Additional UIF, matl.uif:</u>

```
$ ****  MATERIALS  ****

MATD 1 CONS
E MU RHO ALPH
16E6 .3 .161 0

MATD  2 ESPE
INCO 718
```

FIGURE 5. KEYWORD INPUT FILE FOR EXAMPLE BYPASS FRAME MODEL ALSO SHOWING ADDITIONAL UIF

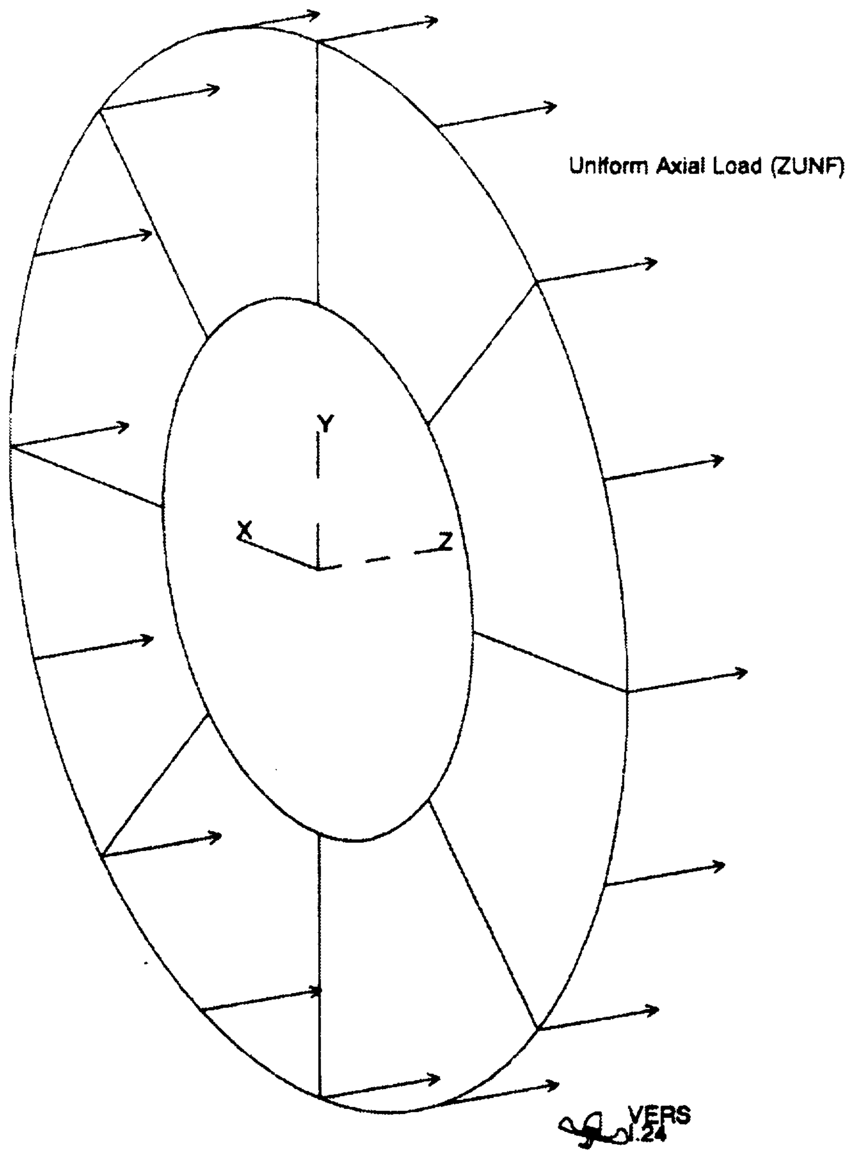


FIGURE 6. Uniform Axial Load (ZUNF)

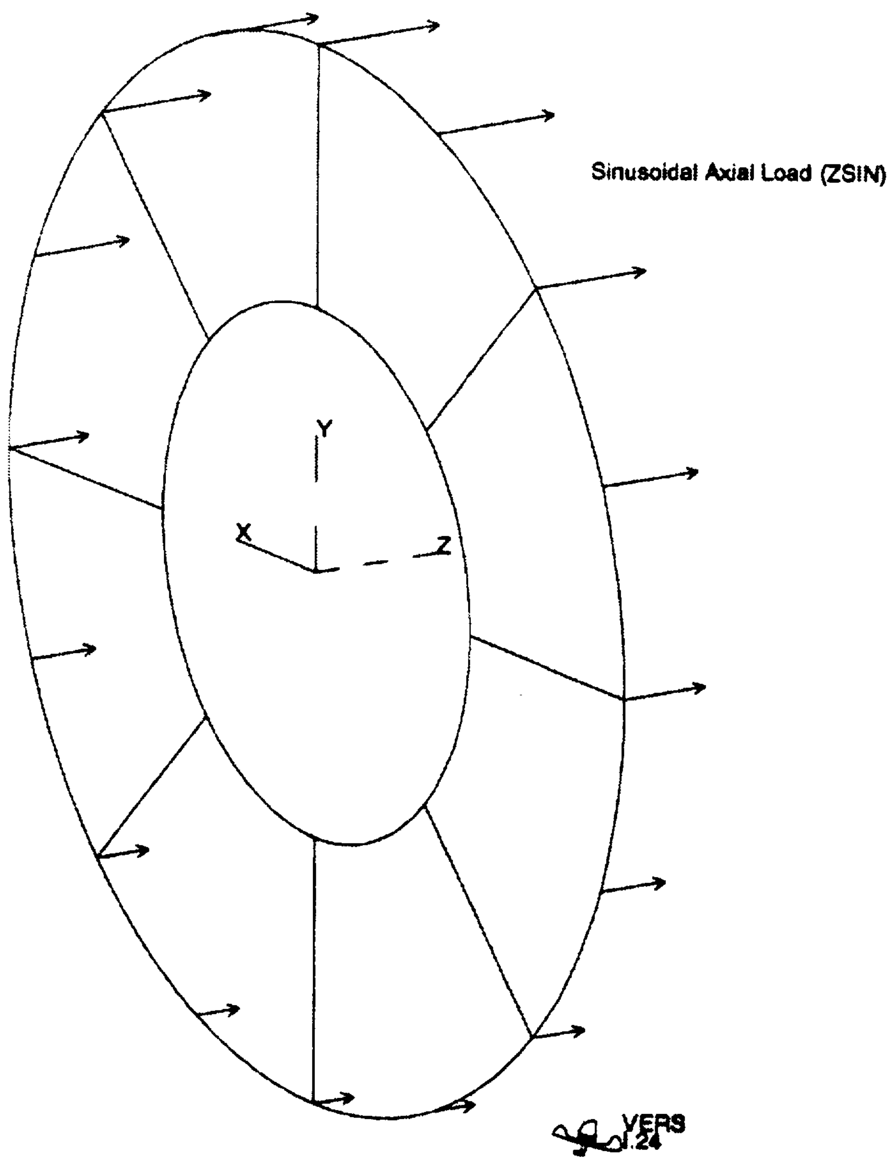


FIGURE 7. Sinusoidal Axial Load (ZSIN) for Phase Angle of 0

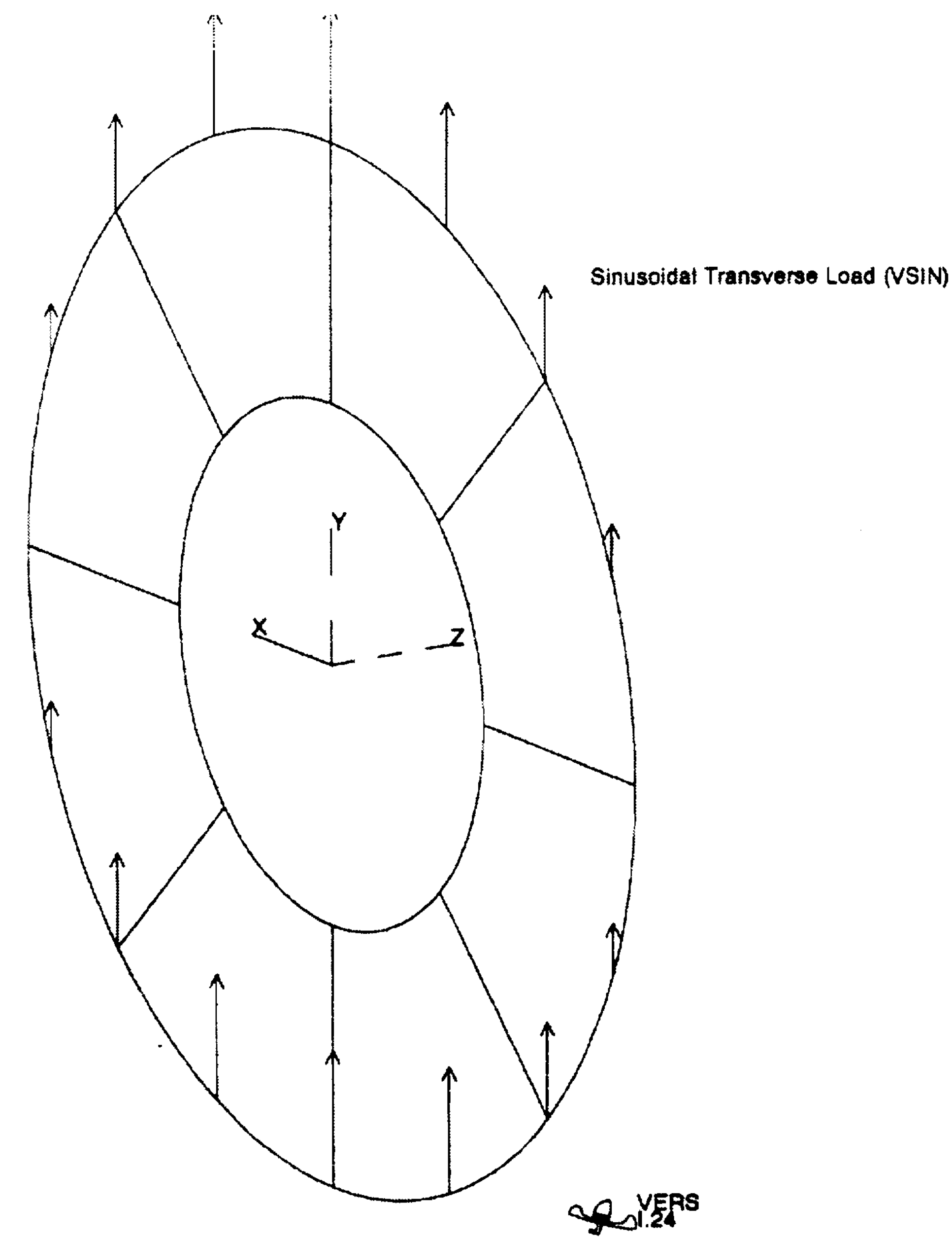


FIGURE 8. Sinusoidal Transverse Load (VSIN) for Phase Angle of 0

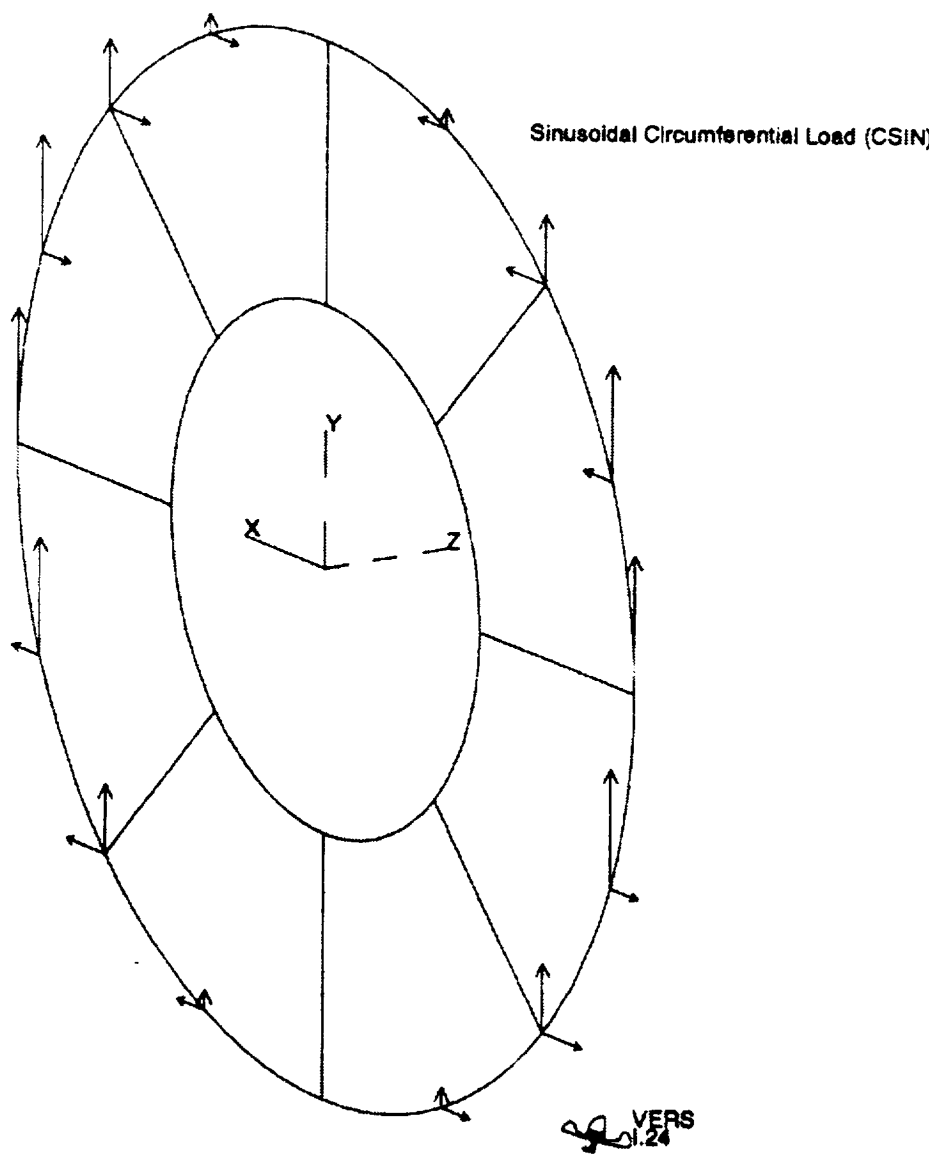


FIGURE 9. Sinusoidal Circumferential Shear Load (CSIN) for Phase Angle of 0

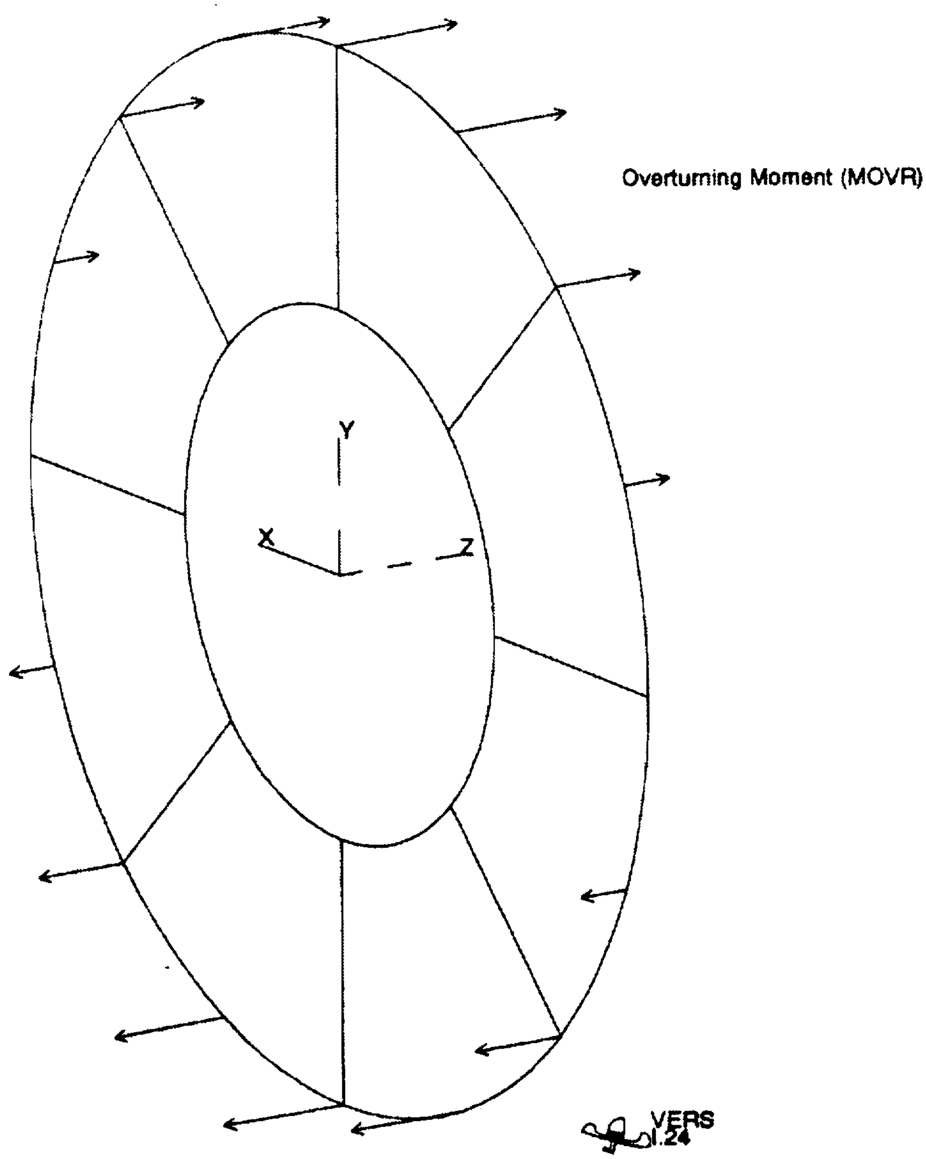


FIGURE 10. Overturning Moment (MOVR) for Phase Angle of 0

APPENDIX D

DISKPC USER'S MANUAL

SIESTA FUNCTION SUMMARY

FUNCTION:
DISKPC

SUB-MENU LOCATION:
Analysis (Sub-Menu 7, Function 8 or **XDISKPC**)

PURPOSE:
Perform elastic, cyclic plasticity and creep analysis for rotating disks.

INPUT FILE(S):
DISKPC input file (02)

OUTPUT FILE(S):
DISKPC output file (04)
DISKPC restart file (07)

REQUIRED USER INPUT:
NONE

COMMENTS:
There is a DISKPC Post Processor available in SIESTA (Sub-Menu 7, Function 9 or **DPCPP**) which generates DISKPC single case output and mission output data plots.

FUNCTION VERSION INFORMATION:
Any changes made to this function after version 2.3.0 07-95 will not be reflected in this release of the manual.

DISKPC USERS' MANUAL - 1995 REVISION

The DISKPC program provides the engineer with a fast, economical tool for performing cyclic inelastic analyses of disks with thermal, centrifugal, and radial mechanical loadings. This document explains the use of the program, the method of running the program, and includes example problems. The disk is analyzed by assuming a condition of axisymmetric plane stress to exist. The plasticity analysis is performed by the method of subvolumes. For creep analyses, time-hardening, strain-hardening, and life-fraction rules are available. Additional capabilities include: small or large displacement analyses, restart, automatic incrementing of RPM loading, and automatic data generation. This documentation is based on the original DISKPC manual, reference (19), by Peter Chen, Jim Brown, and Richard McKnight. This revision is by Greg Bechtel.

1.0 SCOPE

1.1 DISCUSSION

The DISKPC computer program discussed in this report was written to give a fast evaluation of rotating disks, under the conditions of linear elasticity and nonlinear cyclic plasticity and creep. The DISKPC program is currently available for use on the HP workstation computers as a part of the SIESTA system.

The theoretical background for the basic finite difference program is contained in references (1) through (5) and further analytical developments of various portions of the program are discussed elsewhere in this report. A previous batch program, reference (6) and a previous time sharing program, reference (7), have also been based on this general method.

1.2 STRUCTURE ANALYZED

The type of structure which is analyzed by this program is a thin disk of arbitrary profile which is symmetrical about its mid-surface and its axis of rotation. The disk may be subjected to any combination of rotational, thermal, and radial forces. There are no restrictions on the disk dimensions as long as the analyst is aware that the solution is based on a plane stress assumption and he evaluates his disk against this assumption. The program has an option to perform large displacements, large strain analyses. This option is user controlled through an input indicator.

1.3 MATERIAL ASSUMPTIONS

The cyclic plasticity analysis performed by this program uses the method of subvolumes. The theoretical development of this technique is detailed in reference (8) and a finite element program, CYANIDE, which uses this technique is covered by reference (9). The temperature dependent stress-strain curves are simulated by multilinear representations which are defined by up to ten stress-strain pairs. The subvolumes method then mathematically manipulates this data such that the Bauschinger effect cross-hardening, and memory are all automatically simulated. These effects are exhibited by real material. For cyclic creep analysis, a "five term" creep equation, *Ec = k*(S **n)*(T**m) + q*(S**r)*T,* is used. In this creep equation, Ec is the equivalent creep strain, S is the normalized von Mises effective stress (psi/100000), T is the time in hours and k, n, m, q, and r are temperature variable constants. Three creep rules are operational in the program; time hardening, strain hardening, and life fraction. Both stress and temperature cutoffs are available below which zero creep is assumed to occur.

1.4 CREDIBILITY

The finite difference solution of disk problems has been used quite extensively since it was first proposed. The first five references give examples of its use. Reference (10) details the use of the method in a situation in which experimental data verified the analytical predictions. This finite difference method is the basis for most of our elastic disk programs. The elastic solution portion of the program has been verified by comparison with theoretical solutions. The plasticity and creep solutions are new additions to the method. The plasticity and creep solutions have been correlated with predictions of published solutions, and with the GEAE nonlinear finite element program, CYANIDE.

1.5 FUTURE DEVELOPMENTS

As of 1978, it was thought that DISKPC had "almost reached its limit of development". At that time the only improvements under consideration were: " allowance for discontinuity in disk geometry, any new creep theories, and post processing for life calculations, multiple material properties". From 1978 until 1995 there was little development activity on DISKPC. In 1995 a number of improvements were made to DISKPC, and a number of additional improvements are planned for the near future. Possible future enhancements include numerous additional pre- and postprocessing capabilities, optimization (combine with the OPTD program), and development of a "ROTORPC" driver program to run "coupled" DISKPC analyses of an entire rotor (include the interactive effects of spacer arms, etc.).

2.0 LIMITATIONS

2.1 PHYSICAL

The analysis assumes a plane stress condition; therefore, any stresses due to variations across the thickness or out of plane loading will not be predicted. The finite difference representation of the disk must be a smooth one, i.e., there can be no jumps in thickness at a radial station. The disk must be of a single material and cycling is assumed to have no effect on the modulus, the coefficient of thermal expansion, and Poisson's ratio. Also, the thermal stress problem is considered to be decoupled, i.e., there is no interdependence between the heat being transferred to and from the disk and the state of strain in the disk.

2.2 COMPUTER

Current program limits may be obtained by running DISKPC, and entering .lim when prompted for the input file name. These limits can be increased if necessary, although model refinement beyond these limits may be overkill, considering the physical assumptions inherent in a DISKPC model.

3.0 PROCEDURE

3.1 MODELING

The disk is modeled by specifying stations along a radial line. The disk is symmetric about this radial line. The first station is at the center of a solid disk or at the bore of an annular disk. The stations are then numbered consecutively outward to the rim. Two pieces of disk geometry data (radius and thickness) are required at each station. There is a payoff to be obtained in computational time by making the station spacing (radial distance between stations) as uniform as possible. However, a finer station spacing should be used in areas of sharp geometric change or in areas where large strain gradients are expected. The analyst will gain insight into the proper station spacing from experience with the program. Figure 1 shows the disk modeling parameters.

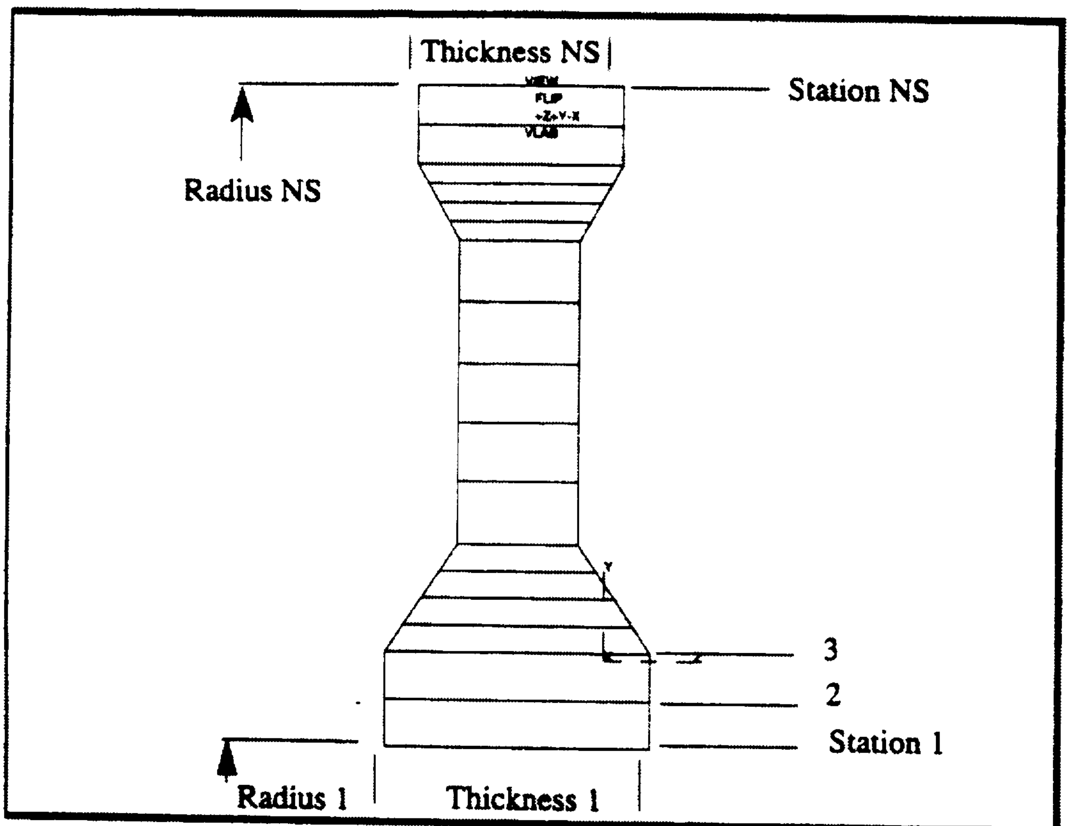


Figure 1

3.2 INPUT SUMMARY

This section gives a brief description of the input to the program. Additional description of the input may be found in the next section. There are thirteen different "groups" of data that may be used in a DISKPC run.

Note that line numbers are required, but are not used by the program. Also note that some clarifying comments are added to the right of the input data (after a "-"), but these comments are not allowed in the actual input file.

I. Overall Control Data:

Line# NS NP NC NCY RTEM NMT NXX MD NANALY NDISP NRESTA

where NS = Number of stations
NP = Number of load conditions (per cycle)
NC = Number of mission cycles
NCY = Number of revised cycles
RTEM = Reference temperature (F)
NMT = Number of temperatures for elastic (and plastic) material properties
NXX = Code for disk geometry (1=solid disk, 2=annular disk)
MD = Number of stations having initial strains specified
NANALY = Analysis code (=0 for elastic, 1 for plastic, 2 for plastic and creep)
NDISP = Deformation Code (=0 for small deformation, =1 for large deformation)
NREST = Restart Output Code (=0 for no restart output, =1 for output)

II. Disk Geometry (repeat until NS stations are defined)

Line# STA RAD THK - Disk geometry for first station (bore i.d.)
Line# STA RAD THK - Disk geometry for next station
......
Line# STA RAD THK - Disk geometry for last station (STA=NS)

Note that a "generation process" will be used if a minus sign is input with the station number. If generation occurs, a linear interpolation scheme is used between the previously input station number and this station number with an increment of 1 station.

For example, the following input:

10	1	1	1
20	-5	2	3
30	-7	4	5

would generate the following values:

Station Number	Radius	Thickness
1	1.00	1.0
2	1.25	1.5
3	1.50	2.0
4	1.75	2.5
5	2.00	3.0
6	3.00	4.0
7	4.00	5.0

III. Temperatures (F) for Material Properties (elastic and plastic, input NMT temperatures)

Line# TEMP#1 TEMP#2 TEMP#3 .... TEMP#NMT

IV. Elastic Material Properties (repeat for NMT temperatures)

Line# RHO E NU ALPHA - elastic data at temp#1
Line# RHO E NU ALPHA - elastic data at temp#2
.....
Line# RHO E NU ALPHA - elastic data at temp#NMT

where RHO = weight density of the material (lbs/in^3, typically @0.300)
E = Young's Modulus (Msi, i.e. input 30. for a modulus of 30E6 psi)
NU = Poisson's ratio
ALPHA = Thermal expansion coefficient
(units are 1E-6 in/in/F, i.e. input 6. for 6.E-6 in/in/F)

V. Plasticity Control Data (skip if NANALY = 0)

Line# NPS MM TOL NPCODE

where NPS = Number of points defining each stress-strain curve (same # for each curve)
MM = Max number of interations for convergence (typical value = 50)
TOL = Plastic strain convergence tolerence (typical value = 1.E-7 inches/inch)
NPCODE = Action code for non-convergence (=0 to continue, =1 to stop program)

VI. Plastic Material Properties (skip if NANALY = 0, repeat for NPS points*NMT temps)
(stress and strain at the points on the stress-strain curves at the corresponding temperatures)

Line# Stress (psi) Strain (inches/inch) - point 1, temperature 1
Line# Stress (psi) Strain (inches/inch) - point 2, temperature 1
.....
Line# Stress (psi) Strain (inches/inch) - point NPS, temperature NMT

Note: For the first point on each curve, E = (stress/strain)

VII. Creep Control Data (skip if NANALY is not 2)

Line# NMTC MMC TOLC LAWCRP NRUPT NRUPS

where NMTC = Number of temperatures for creep constants
MMC = Number of increments for each creep time
TOLC = Temperature (F) below which creep will be ignored
LAWCRP = Creep hardening rule (=0 for time, =1 for strain, = 2 for life fraction)
NRUPT = Number of rupture data temperatures
NRUPS = Number of rupture stresses per rupture data temperature

VIII. Temperatures (F) for Creep Constants (skip if NANALY is not 2, input NMTC temps.)

Line# TEMP#1 TEMP#2 TEMP#3 .... TEMP#NMTC

IX. Creep Constants (skip if NANALY is not 2, input NMTC lines of constants)

Line# m n k q r sigcut - constants and cutoff stress at temp #1
Line# m n k q r sigcut - constants and cutoff stress at temp #2
......
Line# m n k q r sigcut - constants and cutoff stress at temp #NMTC

Note: A "five term" creep equation, *Ec = k*(S **n)*(T**m) + q*(S**r)*T,* is used. In this creep equation, Ec is the equivalent creep strain, S is the normalized von Mises effective stress (psi/100000), T is the time in hours and k, n, m, q, and r are temperature variable constants. The cutoff stress, sigcut (psi), is the effective stress below which creep is ignored.

X. Rupture Map Data (skip if NANALY is not 2, required if LAWCRP=2)

a. Rupture Temperatures (F) (input NRUPT values)

Line# TEMP#1 TEMP#2 TEMP#3 .... TEMP#NRUPT

b. Rupture Stresses (psi) at corresponding rupture temperatures (repeat for NRUPT temps)

Line#	Stress#1	Stress#2	Stress#3	....	Stress#NRUPS	- at temp #1
Line#	Stress#1	Stress#2	Stress#3	....	Stress#NRUPS	- at temp #2
....						
Line#	Stress#1	Stress#2	Stress#3	....	Stress#NRUPS	- at temp #NRUPT

c. Rupture Times (hours) at corresponding stresses and temperatures (repeat for NRUPT temps)

Line#	Time#1	Time#2	Time#3	....	Time#NRUPS	- at temp #1
Line#	Time#1	Time#2	Time#3	....	Time#NRUPS	- at temp #2
....						
Line#	Time#1	Time#2	Time#3	....	Time#NRUPS	- at temp #NRUPT

<u>XI. Cycle Revision Data</u> (skip if no cycle revision, i.e. NCY=0; repeat for NCY cycle revs.)

Line#	NCR	NREV	LCSA	LCSB	LCSC ... etc.	- cycle revision input for first revised cycle
Line#	NCR	NREV	LCSA	LCSB	LCSC ... etc.	- cycle revision input for second revised cycle
.....						
Line#	NCR	NREV	LCSA	LCSB	LCSC ... etc.	- cycle revision input for last revised cycle

XII. Initial Strain Data (skip if no initial strains, i.e. skip if MD=0)

a. Total Physical Time (hours) at which corresponding initial strains are given (not necessarily creep time)

Line# TOT

b. Initial Strains (input one line each for MD stations, four strain values per station)

Line# STA Eps-Plas-Rad Eps-Plas-Hoop Eps-Creep-Rad Eps-Creep-Hoop

c. Accumulated Creep Time (hours), Eff. Creep Strain (inches/in), and Life Fraction (decimal) (again, input one line each for MD stations, even if there is no creep in this analysis)

Line# STA CreepTime Eps-Creep-Eff FRACT

XIII. Load Condition Data (this data group is repeated for NP load cases)

a. Overall Load Condition Data

Line# SIG-RIM SIG-BORE RPM NTEMP DTIME NFORCE NINC

where: SIG-RIM = rim stress (psi), positive if in radial tension
SIG-BORE = bore stress (psi), positive if in radial tension
RPM = rotation speed (obviously in revolutions per minute)
NTEMP = Number of stations whose temperatures differ from previous load case
DTIME = Time increment for creep (hours)
NFORCE = Number of stations whose concentrated forces are changed for this case
NINC = Number of increments from previous rpm to this rpm

b. Temperatures Changed (skip if no temp changes, i.e. skip if NTEMP=0; input NTEMP lines)
Note: Linear interpolation generation is triggered if station number is input as negative

Line#	STA	TEMP(F)	- first line of temperature revision data
Line#	STA	TEMP(F)	- second line of temperature revision data
....			
Line#	STA	TEMP(F)	- NTEMP line of temperature revision data

c. Concentrated Forces Changed (skip if no changes, i.e. NFORCE=0; input NFORCE lines)
Note: Linear interpolation generation scheme is triggered if station number is input as negative
Also Note: Concentrated Force is based on the complete circle (i.e. total pounds)

Line#	STA	FORCE(LB)	- first line of force revision data
Line#	STA	FORCE(LB)	- second line of force revision data
....			
Line#	STA	FORCE(LB)	- NFORCE line of force revision data

3.3 DISCUSSION OF INPUT

The input to the program is by means of a line numbered data file. Each line of data and the order of the lines are described below. Each data entry is separated from the adjacent entry by one or more consecutive blank spaces or by a single comma (,). Each and every comma (,) is counted as a data field delimiter. One or more consecutive blank spaces is counted as a data field delimiter. It is recommended that blanks be used for visual legibility.

As shown on the input summary sheets, the input data are broken into 13 categories, some of which contain several sub-groups. The analysis may not require all the categories or groups. NOTE: the line numbers are not part of the data but are for the user's information only.

An optional data generation scheme, using a linear interpolation method, is available in the areas of (1) disk geometry in Category II, (2) station temperature in Category XIII, and (3) station concentrated forces in Category XIII. The generation scheme is illustrated in Category II of input summary section.

Even though the program utilizes either bilinear interpolation or extrapolation of temperatures, material properties, plastic stress-strain relationships, and creep data, etc., it is recommended that the given data should be input over a broad enough range so that the extrapolation process can be avoided. Previous experiences have shown that the extrapolation scheme may result in inaccurate properties which will in turn yield misleading solutions.

The input data forms are designed such that almost all the input items are self-explanatory. Only those needing further discussion are described below:

<u>Category I (Overall Control Data)</u>:

NC Number of repeated mission cycles to be analyzed. If NC is input as negative, then the default is changed to <u>not</u> print out results and the user inputs cycle numbers to print out on second line of the file.

NP Number of load conditions per mission cycle.

NCY Number of revised cycles. This is designed to serve two purposes:
(i) Deleting of some load conditions within the mission cycle.
(ii) Suppressing output prints for some load conditions within the mission cycle.

NMT Number of temperatures for the given material properties.

The program uses a bilinear interpolation or extrapolation technique to evaluate material properties as a function of temperatures. Generally, NMT will be greater than or equal to 2. However, NMT can be equal to 1 if a sole temperature condition exists.

NANALY	If 0, elastic analysis and skip plasticity data. If less than 0, elastic analysis but read plasticity data.
NDISP	If 0, small deformation analysis. If 1, large deformation analysis (update geometry each load step).

If permanent plastic strains occurred in the loading paths, the virgin disk geometry is assumed to remain the same for small deformation analysis, but varies at beginning of each load condition for large deformation analysis.

NRESTA A code to identify the request of creating a restart file.

Restart capability saves the up-to-date strains and their related information in the file 07. This stored information can then be used, with minor modifications, for continuing the load path if requested in another run. Because DISKPC runs so quickly, this feature is not generally used.

Category V (Plasticity Control Data):

NPS Number of stress-strain points per stress-strain curve.

Each curve is for one temperature. The first stress-strain point is located at the elastic limit of the corresponding temperature, such that: E = sig1/eps1

MM Maximum iterations allowed for plasticity convergence computation. (i.e. MM = 50)

TOL Plasticity convergence strain tolerance. (suggested value TOL = 1.E-7)

The analysis may require a large number of iterations to converge if the station spacing is highly irregular and/or large amounts of plastic flow occur. The user is provided with certain tools to counter this problem. The "TOL" input allows the user to specify the lower bound below which plastic flow is considered to be negligible. The "MM" input allows the user to specify the maximum number of iterations the computer will perform without converging within the specified "TOL" tolerance. "#ITERATIONS", the number of iterations which the computer performed, is an output. This allows the user to make a judgment as to the acceptability of the loading increment which he inputs. If a given loading requires a large number of iterations to converge, it should be rerun using a number of loading increments.

NPCODE A code to provide the execution function described below:

When the number of iterations reaches MM but the convergence tolerance, TOL, is still not met, then continued execution of the rest of the program is accomplished if NPCODE = 0 or stop execution if NPCODE = 1.

Category VI (Plastic Material Properties):

The nominal stress, s0, is defined as the load divided by the original cross-sectional area of the bar. The conventional or engineering strain, e0, is defined as the extension per unit original length. The commonly available stress-strain data from handbooks or tensile tests are based on s0 and e0.

The true stress-strain curve is used for large plasticity cases. The true stress, s, is the load divided by the current cross-sectional area and can be calculated according to the equation: $s = s0\,(1 + e0)$.

The true strain can thus be evaluated by the formula: $e = \ln(1 + e0)$, where ln is the natural log notation.

There will be little difference between engineering strain-stress (s0, e0) and true stress-strain (s, e) if no substantial plastic strain occurs. However, if substantial amounts of plastic strain exists, we recommend the use of true stress-strain curve.

Category VII, VIII, IX (Creep Input):

MMC is the number of increments for each creep time (TCRP) supplied in Category XIII.

The purpose of this data is to subdivide the creep time into smaller time increments for creep computation, i.e., dt = TCRP/MMC. For large values of MMC, the solution will be more accurate, but more computation steps are required. The value of MMC should be at least equal to 1. Suggested value of MMC is at least 10. Of course, a reasonable value for MMC is directly dependent on the time step for the load condition (TCRP).

The three creep rules available for creep analysis are (1) time- hardening, (2) strain-hardening, and (3) life-fraction. The time-hardening rule assumes that the creep rate depends upon the time from the beginning of the creep process. The strain-hardening rule assumes that in going from one stress level to the next, the creep rate depends on the existing strain in the material. The life-fraction rule assumes that the creep rate depends upon the fraction of life used up. The life-fraction creep hardening rule requires that rupture map data be input (Category X). For most disk applications all of these creep hardening rules should give similar results, although the strain-hardening rule is generally recommended.

Category X (Rupture Map Data):

This group of data is geared for the life-fraction rule of creep analysis. The ultimate goal of this rupture map data is to compute the rupture time as a function of temperature and stress, i.e., time = f (temp., stress). The computation requires a 2-dimensional interpolation scheme. Care must be taken that this rupture map data will be wide enough to cover all the actual life-fractions which occur, i.e. avoid extrapolation.

Category XI (Cycle Revision Data):

The word "revision" here means either deleting the specified load condition number (LCN) or suppressing printing results of the specified LCN. The total number of revised cycles given in this category should match the control value of NCY as given in Category I. If Cycle No. Revised (NCR) is positive, then all the specified LCN will be deleted for this NCR. If a minus sign is placed in front of NCR, then this NCR will perform computation but suppress printing the results of all the specified LCN. An automatic generation scheme will be used if a minus sign is put in front of the revision LCN at the ith entry. The generation scheme is such that revisions will be made for all the LCN inclusively from the LCN specified at the (i-l)th entry up to the LCN specified at the ith entry. Total number of LCN entries should be equal to the corresponding NREV at the same data line.

Category XII (Initial Strain Data):

The initial strains, time creeped, eff. strain creeped, and life-time fraction used up are the plasticity and creep values imposed on the structure at the beginning of the problem analysis. This may be used to "restart" an analysis, or to include existing inelastic strains (i.e. residual stress/strain) in the DISKPC analysis.

Category XIII (Load Condition Data):

NTEMP This is the number of line entries of temperatures for this load condition (LCN).

A non-zero value of NTEMP should be used for the first load condition, because station temperatures have not been defined previously.

NINC This is the number of increments between rpm of previous load case and the current case.

For example. if the rpm of the previous load case is rpml, and rpm of current load case is rpm2 and a non-zero NINC is specified, then analysis will be made for a series of rpm's, ranging from (rpm + Drpm) up to rpm2 with increment of Drpm. Drpm is computed according to the formula of Drpm = (rpm2 - rpml)/abs(NINC). If a positive NINC is specified, output for each performed rpm will be printed. If a negative NINC is specified, output for the performed rpm's (except for the final rpm within the range) will not be printed. This piece of information is particularly useful for over speed analysis which may require many rpm increments in order to yield a satisfactory solution. This feature should not be used to increment the speed to zero rpm.

FORCE This is the total external force (in total pounds) acting at the specified station.

A positive value is for a radially outward force. Note that application of a concentrated force will give a stress discontinuity. There is no assurance that DISKPC will output the more critical stress, so it may be a good idea to only apply forces to stations which are very close to adjacent stations (i.e. make a "small" station to apply a concentrated force at).

3.4 INPUT LIMITATIONS AND SUGGESTIONS

Current problem size limitations (as of version 2.3.0 August, 1995):

NS = 100
NP = 100
NC = 100
NMT = 20
NPS = 10
NRUPT = 20
NRUPS = 10
NMTC = 20

Up-to-date program limits may be found by entering ".lim" when prompted by DISKPC for input filename.

Suggested iteration control values:

MM = 50
TOL = 1.0E-6 to 1.0E-7 inch/inch.

3.5 RUNNING THE PROGRAM

The DISKPC program is available on the LEAD Workstations, through the SIESTA family of programs. Type siesta to get into SIESTA. DISKPC is in Sub-Menu 7, Function 8 (7,8), or simply type xdiskpc once you are in SIESTA. The program will ask for the DISKPC input filename. The user then types in his data file name and the program will run. The program output is written on a file, logic unit 04 (f04.dat). Furthermore, if the user has requested restart capability, the restart output will be on a file, logic unit 07 (f07.dat).

3.6 OUTPUT FILES

The regular outputs are written on file f04.dat. The restart outputs are written on file f07.dat The f04.dat output file may be postprocessed with the DISKPC Postprocessor. The DISKPC Postprocessor is available in SIESTA in Sub-Menu 7, Function 9 (7,9), or simply type dpcpp once you are in SIESTA.

4.0 EXAMPLE CASES

4.1 GENERAL COMMENTS

When the original DISKPC manual was written in 1978, the authors pointed out that:

> The program has an extensive history of use already; approximately 300 runs have been made on the program. Many users have correlated the program elastically with other programs, such as FINITE and CLASS/MASS. In addition, the results of the program have been correlated with testing and engine operating results.

Since then DISKPC has been run thousands of times. Most of the test cases listed in the original DISKPC manual, reference (19), will not be included here, but the input files are available as a part of the standard test cases for DISKPC.

As of August 1995, the standard DISKPC test cases are kept on Greg Bechtel's workstation. These files have global read permission, so they can be copied to a users' workstation. There is up to date documentation available in the "readme.doc" file located with these files.

The path to these test cases is "/global/c0405/bechtel/diskpc/newtest". Users may wish to copy any of these files to their own workstation to use as examples.

The Standard Test Cases for DISKPC (as of August, 1995) include:

dpcvt01.inp	check of solid elastic disk equations from DISKPC Manual (R78AEG295) section 4.2, pages 17-20
dpcvt02.inp	check plasticity/creep of an annular disk from DISKPC Manual (R78AEG295) section 4.3, pages 21-29
dpcvt03.inp	elastic annular disk with creep from DISKPC Manual (R78AEG295) section 4.4, pages 30-33
dpcvt04.inp	thermo-elastic-plastic static analys of a thick-walled cylinder - first loadings from DISKPC Manual (R78AEG295) section 4.5, pages 34-37,40 Note: the input listing in the diskpc manaual does not quite work with the current version of DISKPC. (Convergence problems if additional iterations are not used).
dpcvt05.inp	thermo-elastic-plastic analysis of a thick-walled cylinder - second loads from DISKPC Manual (R78AEG295) section 4.5, pages 34,38,39,41 Note: convergence problems in load case 13 unless extra iterations are used.
dpcvt06.inp	creep solution comparison (between DISKPC and ADINA) from DISKPC Manual (R78AEG295) section 4.6, pages 42-45

dpcvt07.inp	thermoplasticity comparison between CYANIDE and DISKPC from DISKPC Manual (R78AEG295) section 4.7, pages 46-50
dpcvt08.inp	check plasticity/creep restart (from dpcvt02.inp) from DISKPC Manual (R78AEG295) section 4.3, pages 21-29
dpcvt08.cmp	baseline comparison to check output for dpcvt08.inp
dpcvt10.inp	test of auto-rpm incrementing run at 50%, 75%, and 100%, then auto rpm back to 50%
dpcvt11.inp	thermoplasticity with full reversed plasticity (extra cases added to dpcvt07.inp input file) Note: DISKPC versions before version 2.3.0 (07-25-95) could not handle this test case properly.
dpcvt12.inp	simple flat elastic annular disk with different rim and bore loadings (pressure and concentrated)
dpcvt13.inp	simple flat disk with different rim, bore, and speed loadings. compare to results of OPTD verification problems 1a, 1b, and 1c.
dpcvt14.inp	disk with variable material properties, etc. compare to results of DISK123 program (used as an OPTD verification problem)
dpcvt15.inp	simple flat disk with intermediate loads compare to ELADI results for OPTD verification

4.2 COMPARISON OF DISKPC WITH THE SOLID ELASTIC DISK EQUATIONS

This example comes from the old DISKPC manual. In this example we demonstrate verification of DISKPC with the theoretical solid disk equations (i.e. a disk with no central hole). These equations are presented in reference (1) and elsewhere.

For the particular case investigated in this example, the following geometry and material properties were used:

```
Ro = 3.0 inches (outside radius)
E  = 30.E6  psi
NU  = 0.3
RHO =  0.283 lb/in3
RPM  = 10000 rpm
```

The DISKPC input file for this problem is as follows:

```
10  16  1  1  0  70  1  1  0  0  0  0
20    1  0.0  0.4
30 -16  3.0  0.4
40  70
50  0.283  30.  0.3  6
60  0  0  10000  2  0  0  0
70     1  70
80  -16  70
```

A sample run of DISKPC using this input follows:

```
 ***************************************************************
      D I S K P C    VERSION 2.3.0  07-26-95
 ***************************************************************
 ENTER INPUT FILE NAME
dpcvt01.inp
 LOGIC UNIT FOR OUTPUT FILE IS 04

 STARTING CYCLE LOOP
  NUMBER OF CYCLES =  1
  NUMBER OF LOAD CASES PER CYCLE =  1

    STARTING ON CYCLE   1
     STARTING LOAD CONDITION   1
```

A listing of the results follows:

D I S K P C VERSION 2.3.0 07-26-95

TOTAL DISK WEIGHT = 3.201 LBS CROSS-SECTION AREA = 1.196 SQ. INCH

INPUT DISK GEOMETRY

STA NO.	RADIUS (INCH)	WIDTH (INCH)
1	.01	.400
2	.200	.400
3	.400	.400
4	.600	.400
5	.800	.400
6	1.000	.400
7	1.200	.400
8	1.400	.400
9	1.600	.400
10	1.800	.400
11	2.000	.400
12	2.200	.400
13	2.400	.400
14	2.600	.400
15	2.800	.400
16	3.000	.400

CYCLE # 1 LOAD # 1
ITERATIONS = 1

SPEED = 10000.000 RPM
RADIAL STRESS (PSI) AT BORE AND RIM ARE .000 .000

ELASTIC-ONLY ANALYSIS

STA NO.	TEMP (DEG F)	FORCE (LBS)	--- STRESSES IN KSI --- SEQ	SR	ST
1	70.	0.	3.001	3.001	3.001
2	70.	0.	2.984	2.980	2.989
3	70.	0.	2.950	2.937	2.962
4	70.	0.	2.896	2.869	2.922
5	70.	0.	2.823	2.775	2.868
6	70.	0.	2.729	2.655	2.798
7	70.	0.	2.617	2.509	2.713
8	70.	0.	2.486	2.336	2.613
9	70.	0.	2.339	2.136	2.498
10	70.	0.	2.176	1.911	2.368
11	70.	0.	2.001	1.658	2.223
12	70.	0.	1.820	1.380	2.062
13	70.	0.	1.639	1.074	1.886
14	70.	0.	1.472	.743	1.695
15	70.	0.	1.339	.385	1.489
16	70.	0.	1.267	.000	1.267

AVERAGE TANGENTIAL STRESS = 2.413E+03 PSI

AVERAGE EFFECTIVE STRESS = 2.291E+03 PSI

STA NO.	DELTA-R (INCH)	--- TOTAL STRAINS IN 1.0E-6 INCH/INCH --- ETEQ	ET-R	ET-T	ET-Z
1	.00000070	87.	70.	70.	-60.
2	.00001397	86.	69.	70.	-60.
3	.00002775	85.	68.	69.	-59.
4	.00004123	84.	66.	69.	-58.
5	.00005427	82.	64.	68.	-56.
6	.00006672	79.	61.	67.	-55.
7	.00007843	76.	56.	65.	-52.
8	.00008926	72.	52.	64.	-49.
9	.00009907	68.	46.	62.	-46.
10	.00010770	63.	40.	60.	-43.
11	.00011502	58.	33.	58.	-39.
12	.00012087	53.	25.	55.	-34.
13	.00012512	47.	17.	52.	-30.
14	.00012761	43.	8.	49.	-24.
15	.00012819	39.	-2.	46.	-19.
16	.00012673	37.	-13.	42.	-13.

The results show excellent correlation with the theoretical equations for radial and hoop stress and radial displacement.

5.0 METHOD OF ANALYSIS & 6.0 NOMENCLATURE

Various equations and derivations for disk analysis, plasticity analysis, and creep analysis are discussed in the original DISKPC Manual, reference (19).

7.0 REFERENCES

1) Manson, S S : Thermal Stress and Low Cycle Fatigue, McGraw-Hill Book Co., Inc., 1966

2) Millenson, M B., and Manson, S. S.: "Determination of Stresses in Gas-Turbine Disks Subjected to Plastic Flow and Creep", NACA Report 906, 1948.

3) Mendelson, A., and Manson, S. S.: "Practical Solution of Plastic Deformation Problems inElastic-Plastic Range", NACA Tech. Note 4088, September 1957.

4) Manson, S. S.: "Analysis of Rotating Disks of Arbitrary Contour and Radial Temperature Distribution in the Region of Plastic Deformation", Proc. 1st National Congr. Appl. Mech., ASME, 1951.

5) Mendelson, A., Hirshberg, M. H., and Manson, S. S.: "A General Approach to the Practical Solution of Creep Problems", J. Basic Engrg., pp. 585-598, December 1959.

6) Moore, C. L., and Cook, W. F.: "Plastic Disk Stress Analysis", R61FPD60, December 20, 1960.

7) McKnight, R. L.: "DISKP-AEG 600 Time Sharing System for Cyclic Plastic Analysis of Rotating Disks", R70AEG284.

8) McKnight, R. L.: "Finite Element Cyclic Thermoplasticity by the Method of Subvolumes", R75AEG477.

9) McKnight, R. L., Brown, J. F., Holt, R. V., and Chen, P. C., "CYANIDE - Cyclic Analysis of Inelastic Deformation - User's Manual", R78AEG285.

10) Mattavi, J. L.: "Low-Cycle Fatigue Behavior Under Biaxial Strain Distribution", ASME Paper No. 68-WA/Met-8.

11) Symonds, P. S.: "Shakedown in Continuous Media", J. of Appl. Mechs., pp. 85-89, March 1951.

12) Budiansky, B.: "A Reassessment of Deformation Theories of Plasticity", J. of Appl. Mechs., pp. 259-264, June 1959.

13) Nadai, A.: Theory of Flow and Fracture of Solids, Vol. II, McGraw-Hill Book Co., Inc., 1963.

14) Mendelson, A.: Plasticity: Theory and Application, MacMillan & Co., New York, 1968.

15) Lubahn, J. D. and Felgar, R. P.: Plasticity and Creep of Metals, John Wiley & Sons, Inc., New York, 1961.

16) Laflen, J. H.: "A Constitutive Theory for Metal Creep", R75AEG505.

17) Bathe, K.J. "ADINA - A Finite Element Program For Automatic Dynamic Incremental Nonlinear Analysis", Report 82448-1, Massachusetts Institute of Technology, May 1977.

18) Finnie, I. and Heller, W.R. Creep of Engineering Materials McGraw-Hill, 1959, p. 208.

19) Chen, P. C, Brown, J. F., and McKnight, R. L., "DISKPC - Timesharing Computer Program for Cyclic Plasticity and Creep Analysis of Rotating Disks - Users' Manual", R78AEG295.

APPENDIX E

T/BEST UPGRADE

Computing Temperature Dependent Metal Properties for T/BEST

By

Galib H. Abumeri
NYMA Inc.
NASA LeRC Group
Brook Park, Ohio

Prepared For

General Electric Aircraft Engines
Cincinnati, Ohio

Contract Number 200-1Q-14N44065
Task # 4162

December 1995

TABLE OF CONTENT

Page

APPENDICES:

FIGURES:

List of Figures (Continued)

1. OBJECTIVE

The objective of this contract is to develop a computational procedure that determines temperature dependent metal properties for the Technology Benefit Estimator code T/BEST [Ref. 1]. The properties generated will be used in the structural analysis of various engine components such as fan, compressor and turbine blades and disks, burner, duct, nozzle, shaft, and inlet. Many of these components are subjected to severe thermal loads increasing the need for materials exhibiting higher strength at progressively higher temperatures. A FORTRAN routine is developed to generate temperature dependent material properties for a variety of nickel-base super alloys. Mechanical, thermal, and creep rupture properties are automatically produced for each component in the engine. The material specification and temperature range are extracted from the T/BEST neutral file.

2. COMPUTATIONAL PROCEDURE

A generic material behavior model (multi-factor interactive equations [Ref. 2]) is used to evaluate the material properties for the components subjected to high temperature environments and high cycle loading conditions. The multi-factor model used to compute mechanical properties such as elastic modulus, yield strength, tensile elongation, and ultimate tensile strength is shown in this equation:

$$M_p = M_{po} \left[(T_M - T)/(T_M - T_0) \right]^n \qquad (1)$$

Where M_p is the material property at current temperature, M_{po} is the reference material property at the reference temperature T_0, and T_M is the material melting temperature. The exponent n can be determined from available experiment data or can be estimated from the anticipated material behavior

When estimating thermal properties, such as thermal expansion coefficient, specific heat, and thermal conductivity, equation (1) may be applied if a negative exponent is used or must be modified as follows :

$$M_p = M_{po} \left[(T_M - T_0)/(T_M - T) \right]^n \qquad (2)$$

The design of components subjected to sustained loading at high temperatures cannot rely on the short-time tensile properties of metals at those temperatures. Elevated temperatures are associated with a mechanical response that causes progressive deformation of a material leading to its rupture (fracture) at a final time t_f. A computational procedure is introduced here to predict the rupture strength at time t. This procedure employs the following form of the multi-factor equation

$$M_p = M_{po} \left[(T_M - T)/(T_M - T_0) \right]^n \left[(1 - (\sigma_t/S_f)(t/t_f) \right]^q \qquad (3)$$

Where σ_t/S_f is the ratio of stress at time t divided by the strength at the final (total) time, t/t_f is the ratio of current time over the final time, and q is the time exponent. Note that equation (3) accounts for two factors: temperature and time. Different numerical values may be assigned to the exponents n and q. A calibration process is required to determine the proper exponents as discussed in section 3.2 of this report.

3. CONSTRUCTION OF MATERIAL DATA BANK

A material data bank containing seven nickel-base super-alloys [Ref. 3] and one titanium alloy [Ref. 4] has been constructed. A complete listing of the material data bank file , *matdata.bank*, is available in Appendix A.

3.1 DESCRIPTION OF MATERIAL DATA BLOCK

Every metal in the material data bank requires a standard set of input information. The input format required is described here. Each material is identified through a unique Keyword using a format A15. For example, if the material of a compressor blade, is defined in the neutral file as ALLOY 713C, then to generate temperature dependent properties for this component, the same keyword must be available in the data bank. A line beginning with a '$' sign is used for comments. A sample data block for a typical metal with the required order is shown below:

Typical Material Data Block From *matdata.bank*

```
$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY  713C (Ref. INCO)
ALLOY 713C                          (Material Keyword, format A15)
WEIGHT DENSITY                      RHOM       0.28600E+00    lb/in**3
RHOM EXPONENT                       EXPRH      0.00000E+00    non-dim
ELASTIC MODULUS                     EM         0.29900E+08    psi
EM EXPONENT                         EXPEM      0.24499E+00    non-dim
POISSON"S RATIO                     NUM        0.30000E+00    non-dim
NUM EXPONENT                        EXPNU      0.00000E+00    non-dim
THERMAL EXPANSION COEFF.            ALPHAM     5.44500E-06    in/in/F
ALPHA EXPONENT                      EXPAL      0.34999E+00    non-dim
HEAT CONDUCTIVITY                   KM         0.70150E+02    BTU/ft**2/in/hr/F
KM EXPONENT                         EXPKM      0.57999E+00    non-dim
SPECIFIC HEAT                       CM         0.10000E+00    BTU/lb/F
CM EXPONENT                         EXPCM      0.32999E+00    non-dim
ULTIMATE TENSILE STRENGTH           STUM       0.12300E+06    psi
STUM EXPONENT                       EXPST      0.50000E+00    non-dim
YIELD STRENGTH (0.2% OFFSET)        SYM        0.10700E+06    psi
SYM EXPONENT                        EXPSY      0.29999E+00    non-dim
REFERENCE RUPTURE STRENGTH          RUP0       0.12300E+06    psi
RUPTURE STRENGTH TEMP. EXPO.        RTPEXP     0.35000E+00
RUPTURE STRENGTH TIME  EXPO.        RTIEXP     0.15000E+01
TENSILE ELONGATION                  EPSTUM     0.80000E+01    %
EPSTUM EXPONENT                     EXPEP      0.50000E+00    non-dim
METAL REFERENCE TEMPERATURE         TREF       0.70000E+02    F
PROPERTY TEMP. INCREMENT            DELTAT     0.50000E+02    F
METAL MELTING TEMPERATURE           TMELTM     0.23500E+04    F
```

The data block shown here repeats for every metal in the data bank. Each entry is composed of a brief identification of the property followed by a keyword and a numerical value with format 40x,E12.5.

The mechanical properties generated at various temperatures are: density, poisson's ratio, elastic modulus, shear modulus, ultimate strength, yield strength, and tensile elongation. The thermal properties are: thermal expansion coefficient, thermal conductivity, and specific heat. The parameter DELTAT represents the temperature increment at which properties are generated. The temperature range varies from the reference temperature to the maximum operating one. The creep rupture strength properties at time t are automatically generated for all metals in the data bank for the specified temperature range.

3.2 CALIBRATION OF ALLOYS

The multi-factor interactive model provides a good prediction of mechanical and thermal properties at higher temperatures when a proper exponent is used. The exponent may vary from one property to the other and is normally obtained through experiments. Since the literature used here provided some data for several properties at higher temperatures, a special programming procedure was employed to determine the exponents which provide properties that match closely the published data. This procedure was executed for all properties of each metal with the exception of density and poisson's ratio which are presumed not to vary (for now) with temperature.

For the metals listed in the data bank (Appendix A), the proper exponents were determined for the following properties: elastic modulus, thermal expansion coefficient, specific heat, and thermal conductivity. Figure 1 shows the calibration of nickel base alloy 713C for the elastic modulus which is plotted as a function of temperature. The elastic modulus predicted by the multi-factor interactive model using an exponent of 0.245 is very close to the excat one. Figures 2, 3 and 4 show respectively the calibration of nickel base alloys IN-100, MAR-M 200 and IN-100 for mean coefficient of thermal expansion, thermal conductivity and specific heat. The properties predicted by the multi-factor equation are in very good agreement with the published data.

Calibration carried out for the tensile elongation, yield strength, and ultimate tensile strength of the nickel alloys indicate that these properties are bi-modal. The multi-facor equation predicts well the behavior of bi-modal functions if two exponents are used. Figure 5 shows that the tensile elongation of Alloy B-1900 is predicted with great accuracy by the multi-factor equation when using an exponent of 0.135 for a temperature up to 1400 °F and an exponent of -0.125 for higher temperatures. Note that there is no change in the tensile elongation when going from a temperature of 1400 °F to 1600°F. The literature [Ref. 3] did not provide a value for the tensile elongation at 1500 °F which would be lower than the one given at 1400°F. Calibration carried out for the yield strength and ultimate strength of Alloys B-1900 and IN-100 indicated that these properties were bi-modal requiring two exponents to best estimate their behavior at elevated temperatures (Figures 6 and 7).

The prediction of rupture strength at time t requires the use of two terms in the multi-factor equation with one exponent each. Calibration was performed for the nickel alloy B-1900. The results shown in Figure 8 indicated that the predicted rupture strength compared well with the experimental one when exponents of 0.38 and 1.55 were used for temperature and time respectively. Figure 8 shows the rupture strength divided by the room temperature reference strength versus the ratio of actual time divided by the final time at several temperatures (70, 500, 1000, 1500, and 2000 °F). If the final time is 1000 hr, the 1500 °F rupture strength at 100 hr can be obtained by reading from the graph in Figure 8 at t/t_f equals 0.1. The results presented in Figure 8 are based on a sustained stress $(\sigma(t)/S_f)$ of 90%. Note that the rupture strength decreases rapidly as the ratio $(\sigma(t)/S_f)$ approaches 1.

4. DESCRIPTION OF PROPERTY GENERATOR MODULE

A FORTRAN module named *propgen.f* is developed to generate temperature dependent properties. A listing of *propgen.f* is available in Appendix B. The property generator routine processes the neutral file of T/BEST and extract the following parameters for two groups of components:

a) Group I. This contains each stage of fan (FAN), compressor (HPC and LPC), and turbines (HPT and LPT) blades and disks. The blade material is identified by the keyword BLDMAT and the disk material is defined as DISMAT. The temperature at the entrance and the exit of the component are stored under the keywords STTINP and STTOUT respectively. Once the temperature range and the material type for a blade are extracted from the T/BEST neutral file, the material data bank file *matdata.bank* is used in matching the correspondent material. Then *propgen.f* generates properties for the selected component from the material reference temperature to the maximum operating temperature with an increment of DELTAT. Note that the temperature increment is defined in the material data bank file. Tables of properties at various temperatures are generated automatically and written to a file named *properties.out*.

b) Group II. The second group includes the following components: burner, inlet, shaft, duct, nozzle, augmenter and mixer. The material type and temperature are defined under the keywords CMPMAT and CTEMP respectively. As in Group I, the component properties are generated from the reference temperature to CTEMP with an increment of DELTAT. Again, the properties generated are stored in the file *properties.out*.

A section of the T/BEST neutral file showing the various parameters that are used by *propgen.f* is listed in Appendix C. Also, a section of the output file *properties.out* is listed in Appendix D. The property generator module can be used as a part of COSMO (Component Specific Modeling Software) to feed the CSTEM input files with the properties required for structural analysis. Also, *propgen.f* can be used as a stand alone routine to generate temperature dependent material properties. Note that the material data bank file, *matdata.file*, is a generalized data bank and a user may add his/her own material data.

Subroutine RUPTURE in the *propgen.f* module predicts the rupture strength for each metal in the material data bank. RUPTURE generates a table of rupture strength versus t/t_f at various temperatures. A sample output is listed at the end of Appendix D.

5. SUMMARY

The results presented in this report demonstrate that the multi-factor interactive equation model is a powerful methodology to accurately evaluate metal properties at high temperatures. With only three parameters, reference property, temperature and exponent, one can easily determine the value of a selected property at any temperature. Employing the methodology presented here helps reducing costly experimental procedures. The material databank developed under this contract contains reference properties for metals (nickel alloys and titanium) that are used in typical aerospace applications. With minimum effort, a user can add more metals to the material data bank.

6 REFERENCES

1. Generazio, E.R., Chamis, C.C., and Abumeri, G.H: Technology Benefit Estimator (T/BEST) User's Manual, NASA TM 106785, December 1994.

2. Chamis, C.C., Structures Division, NASA Lewis Research Center, Cleveland, Ohio.

3. The International Nickel Co. (INCO): High Temperature High Strength Nickel Base Alloys, New York, 1987.

4. Brown, W.F., Mindlin, H., and Ho, C.Y.: Aerospace Structural Metals Handbook, Volume 4, 1995 Edition, published by CINDA/USAF CRDA Handbooks Operation Purdue University.

APPENDIX A: LISTING OF *matdata.bank* FILE

A listing of the material data bank file that has been constructed to be used in conjunction with the T/BEST software is enclosed in this Appendix.

```
METAL PROPERTIES

$ METAL: TITANIUM  (USE FOR APPLICATIONS WITH TEMPERATURE UP TO 750F)
$ REFERENCE: AEROSPACE STRUCTURAL METALS HANDBOOK, 1995 EDITION
Ti-4Al-4Mn
WEIGHT DENSITY                   RHOM      0.16300E+00   lb/in**3
RHOM EXPONENT                    EXPRH     0.00000E+00   non-dim
ELASTIC MODULUS                  EM        0.16000E+08   psi
EM EXPONENT                      EXPEM     0.79490E+00   non-dim
POISSON"S RATIO                  NUM       0.32000E+00   non-dim
NUM EXPONENT                     EXPNU     0.00000E+00   non-dim
THERMAL EXPANSION COEFF.         ALPHAM    4.75000E-06   in/in/F
ALPHA EXPONENT                   EXPAL     0.25990E+00   non-dim
HEAT CONDUCTIVITY                KM        0.55000E+02   BTU/ft**2/in/hr/F
KM EXPONENT                      EXPKM     0.99490E+00   non-dim
SPECIFIC HEAT                    CM        0.12500E+00   BTU/lb/F
CM EXPONENT                      EXPCM     0.67990E+00   non-dim
ULTIMATE TENSILE STRENGTH        STUM      0.15000E+06   psi
STUM EXPONENT                    EXPST     0.50000E+00   non-dim#
YIELD STRENGTH (0.2% OFFSET)     SYM       0.13000E+06   psi
SYM EXPONENT                     EXPSY     0.99400E+00   non-dim#
REFERENCE RUPTURE STRENGTH       RUP0      0.15000E+06   psi
RUPTURE STRENGTH TEMP. EXPO      RTPEXP    0.33000E+00   non-dim#
RUPTURE STRENGTH TIME  EXPO      RTIEXP    0.14400E+01   non-dim#
TENSILE ELONGATION               EPSTUM    0.10000E+02   %
EPSTUM EXPONENT                  EXPEP     0.50000E+00   non-dim#
METAL REFERENCE TEMPERATURE      TREF      0.70000E+02   F
PROPERTY TEMP. INCREMENT         DELTAT    0.50000E+02   F
METAL MELTING TEMPERATURE        TMELTM    0.28200E+04   F

$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY  713C
$ SOURCE: INTERNATIONAL NICKEL CO.
ALLOY 713C
WEIGHT DENSITY                   RHOM      0.28600E+00   lb/in**3
RHOM EXPONENT                    EXPRH     0.00000E+00   non-dim
ELASTIC MODULUS                  EM        0.29900E+08   psi
EM EXPONENT                      EXPEM     0.24499E+00   non-dim
POISSON"S RATIO                  NUM       0.30000E+00   non-dim
NUM EXPONENT                     EXPNU     0.00000E+00   non-dim
THERMAL EXPANSION COEFF.         ALPHAM    5.44500E-06   in/in/F
ALPHA EXPONENT                   EXPAL     0.34999E+00   non-dim
HEAT CONDUCTIVITY                KM        0.70150E+02   BTU/ft**2/in/hr/F
KM EXPONENT                      EXPKM     0.57999E+00   non-dim
SPECIFIC HEAT                    CM        0.10000E+00   BTU/lb/F
CM EXPONENT                      EXPCM     0.32999E+00   non-dim
ULTIMATE TENSILE STRENGTH        STUM      0.12300E+06   psi
STUM EXPONENT                    EXPST     0.50000E+00   non-dim@
YIELD STRENGTH (0.2% OFFSET)     SYM       0.10700E+06   psi
SYM EXPONENT                     EXPSY     0.29999E+00   non-dim@
REFERENCE RUPTURE STRENGTH       RUP0      0.12300E+06   psi
RUPTURE STRENGTH TEMP. EXPO      RTPEXP    0.38000E+00   non-dim@
RUPTURE STRENGTH TIME  EXPO      RTIEXP    0.15500E+01   non-dim@
TENSILE ELONGATION               EPSTUM    0.80000E+01   %
```

```
EPSTUM EXPONENT                  EXPEP      0.50000E+00    non-dim@
METAL REFERENCE TEMPERATURE      TREF       0.70000E+02    F
PROPERTY TEMP. INCREMENT         DELTAT     0.50000E+02    F
METAL MELTING TEMPERATURE        TMELTM     0.23500E+04    F

$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY 713LC
$ SOURCE: INTERNATIONAL NICKEL CO.
ALLOY 713LC
WEIGHT DENSITY                   RHOM       0.28900E+00    lb/in**3
RHOM EXPONENT                    EXPRH      0.00000E+00    non-dim
ELASTIC MODULUS                  EM         0.28600E+08    psi
EM EXPONENT                      EXPEM      0.26999E+00    non-dim
POISSON"S RATIO                  NUM        0.30000E+00    non-dim
NUM EXPONENT                     EXPNU      0.00000E+00    non-dim
THERMAL EXPANSION COEFF.         ALPHAM     4.69000E-06    in/in/F
ALPHA EXPONENT                   EXPAL      0.57999e+00    non-dim
HEAT CONDUCTIVITY                KM         0.67500E+02    BTU/ft**2/in/hr/F
KM EXPONENT                      EXPKM      0.62499E+00    non-dim
SPECIFIC HEAT                    CM         0.10500E+00    BTU/lb/F
CM EXPONENT                      EXPCM      0.31999E+00    non-dim
ULTIMATE TENSILE STRENGTH        STUM       0.13000E+06    psi
STUM EXPONENT                    EXPST      0.50000E+00    non-dim@
YIELD STRENGTH (0.2% OFFSET)     SYM        0.10900E+06    psi
SYM EXPONENT                     EXPSY      0.33999E+00    non-dim@
REFERENCE RUPTURE STRENGTH       RUP0       0.13000E+06    psi
RUPTURE STRENGTH TEMP. EXPO      RTPEXP     0.38000E+00    non-dim@
RUPTURE STRENGTH TIME  EXPO      RTIEXP     0.15500E+01    non-dim@
TENSILE ELONGATION               EPSTUM     0.15000E+02    %
EPSTUM EXPONENT                  EXPEP      0.50000E+00    non-dim@
METAL REFERENCE TEMPERATURE      TREF       0.70000E+02    F
PROPERTY TEMP. INCREMENT         DELTAT     0.50000E+02    F
METAL MELTING TEMPERATURE        TMELTM     0.24100E+04    F

$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY B-1900
$ SOURCE: INTERNATIONAL NICKEL CO.
B-1900
WEIGHT DENSITY                   RHOM       0.29700E+00    lb/in**3
RHOM EXPONENT                    EXPRH      0.00000E+00    non-dim
ELASTIC MODULUS                  EM         0.31000E+08    psi
EM EXPONENT                      EXPEM      0.24499E+00    non-dim
POISSON"S RATIO                  NUM        0.30000E+00    non-dim
NUM EXPONENT                     EXPNU      0.00000E+00    non-dim
THERMAL EXPANSION COEFF.         ALPHAM     6.38500E-06    in/in/F
ALPHA EXPONENT                   EXPAL      0.20500E+00    non-dim
HEAT CONDUCTIVITY                KM         0.65000E+02    BTU/ft**2/in/hr/F
KM EXPONENT                      EXPKM      0.72900E+00    non-dim
SPECIFIC HEAT                    CM         0.10000E+00    BTU/lb/F
CM EXPONENT                      EXPCM      0.31990E+00    non-dim
ULTIMATE TENSILE STRENGTH        STUM       0.14100E+06    psi
STUM EXPONENT                    EXPST      0.42900E+00    non-dim@
YIELD STRENGTH (0.2% OFFSET)     SYM        0.12000E+06    psi
SYM EXPONENT                     EXPSY      0.47500E+00    non-dim
REFERENCE RUPTURE STRENGTH       RUP0       0.14100E+06    psi
RUPTURE STRENGTH TEMP. EXPO      RTPEXP     0.38000E+00    non-dim
RUPTURE STRENGTH TIME  EXPO      RTIEXP     0.15500E+01    non-dim
TENSILE ELONGATION               EPSTUM     0.80000E+01    %
EPSTUM EXPONENT                  EXPEP      0.13500E+00    non-dim
METAL REFERENCE TEMPERATURE      TREF       0.70000E+02    F
```

```
PROPERTY TEMP. INCREMENT      DELTAT     0.50000E+02    F
METAL MELTING TEMPERATURE     TMELTM     0.23750E+04    F

$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY   IN-100
$ SOURCE: INTERNATIONAL NICKEL CO.
IN-100
WEIGHT DENSITY                RHOM       0.28000E+00    lb/in**3
RHOM EXPONENT                 EXPRH      0.00000E+00    non-dim
ELASTIC MODULUS               EM         0.31200E+08    psi
EM EXPONENT                   EXPEM      0.26999E+00    non-dim
POISSON"S RATIO               NUM        0.30000E+00    non-dim
NUM EXPONENT                  EXPNU      0.00000E+00    non-dim
THERMAL EXPANSION COEFF.      ALPHAM     7.20000E-06    in/in/F
ALPHA EXPONENT                EXPAL      0.18500E+00    non-dim
HEAT CONDUCTIVITY             KM         0.72000E+02    BTU/ft**2/in/hr/F@
KM EXPONENT                   EXPKM      0.81999E+00    non-dim
SPECIFIC HEAT                 CM         0.10000E+00    BTU/lb/F
CM EXPONENT                   EXPCM      0.23499E+00    non-dim
ULTIMATE TENSILE STRENGTH     STUM       0.14700E+06    psi
STUM EXPONENT                 EXPST      0.21000E+00    non-dim@
YIELD STRENGTH (0.2% OFFSET)  SYM        0.12300E+06    psi
SYM EXPONENT                  EXPSY      0.34990E+00    non-dim@
REFERENCE RUPTURE STRENGTH    RUP0       0.14700E+06    psi
RUPTURE STRENGTH TEMP. EXPO   RTPEXP     0.38000E+00    non-dim@
RUPTURE STRENGTH TIME  EXPO   RTIEXP     0.15500E+01    non-dim@
TENSILE ELONGATION            EPSTUM     0.90000E+01    %
EPSTUM EXPONENT               EXPEP      0.31990E+00    non-dim@
METAL REFERENCE TEMPERATURE   TREF       0.70000E+02    F
PROPERTY TEMP. INCREMENT      DELTAT     0.50000E+02    F
METAL MELTING TEMPERATURE     TMELTM     0.24350E+04    F

$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY  IN-162
$ SOURCE: INTERNATIONAL NICKEL CO.
IN-162
WEIGHT DENSITY                RHOM       0.29200E+00    lb/in**3
RHOM EXPONENT                 EXPRH      0.00000E+00    non-dim
ELASTIC MODULUS               EM         0.28500E+08    psi
EM EXPONENT                   EXPEM      0.24499E+00    non-dim
POISSON"S RATIO               NUM        0.30000E+00    non-dim
NUM EXPONENT                  EXPNU      0.00000E+00    non-dim
THERMAL EXPANSION COEFF.      ALPHAM     6.49000E-06    in/in/F
ALPHA EXPONENT                EXPAL      0.28490E+00    non-dim
HEAT CONDUCTIVITY             KM         0.72000E+02    BTU/ft**2/in/hr/F
KM EXPONENT                   EXPKM      0.75000E+00    non-dim
SPECIFIC HEAT                 CM         0.10000E+00    BTU/lb/F
CM EXPONENT                   EXPCM      0.23490E+00    non-dim
ULTIMATE TENSILE STRENGTH     STUM       0.14600E+06    psi
STUM EXPONENT                 EXPST      0.50000E+00    non-dim@
YIELD STRENGTH (0.2% OFFSET)  SYM        0.11800E+06    psi
SYM EXPONENT                  EXPSY      0.20500E+00    non-dim@
REFERENCE RUPTURE STRENGTH    RUP0       0.14600E+06    psi
RUPTURE STRENGTH TEMP. EXPO   RTPEXP     0.38000E+00    non-dim@
RUPTURE STRENGTH TIME  EXPO   RTIEXP     0.15500E+01    non-dim@
TENSILE ELONGATION            EPSTUM     0.70000E+01    %
EPSTUM EXPONENT               EXPEP      0.22490E+00    non-dim@
METAL REFERENCE TEMPERATURE   TREF       0.70000E+02    F
PROPERTY TEMP. INCREMENT      DELTAT     0.50000E+02    F
METAL MELTING TEMPERATURE     TMELTM     0.23800E+04    F
```

```
$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY   MAR-M 200
$ SOURCE: INTERNATIONAL NICKEL CO.
MAR-M 200
WEIGHT DENSITY                  RHOM      0.30800E+00   lb/in**3
RHOM EXPONENT                   EXPRH     0.00000E+00   non-dim
ELASTIC MODULUS                 EM        0.31600E+08   psi
EM EXPONENT                     EXPEM     0.26990E+00   non-dim
POISSON"S RATIO                 NUM       0.30000E+00   non-dim
NUM EXPONENT                    EXPNU     0.00000E+00   non-dim
THERMAL EXPANSION COEFF.        ALPHAM    6.10500E-06   in/in/F
ALPHA EXPONENT                  EXPAL     0.18500E+00   non-dim
HEAT CONDUCTIVITY               KM        0.88000E+02   BTU/ft**2/in/hr/F
KM EXPONENT                     EXPKM     0.81990E+00   non-dim
SPECIFIC HEAT                   CM        0.09500E+00   BTU/lb/F
CM EXPONENT                     EXPCM     0.23499E+00   non-dim
ULTIMATE TENSILE STRENGTH       STUM      0.13500E+06   psi
STUM EXPONENT                   EXPST     0.50000E+00   non-dim@
YIELD STRENGTH (0.2% OFFSET)    SYM       0.12200E+06   psi
SYM EXPONENT                    EXPSY     0.34999E+00   non-dim@
REFERENCE RUPTURE STRENGTH      RUP0      0.13500E+06   psi
RUPTURE STRENGTH TEMP. EXPO     RTPEXP    0.38000E+00   non-dim@
RUPTURE STRENGTH TIME  EXPO     RTIEXP    0.15500E+01   non-dim@
TENSILE ELONGATION              EPSTUM    0.70000E+01   %
EPSTUM EXPONENT                 EXPEP     0.31990E+00   non-dim@
METAL REFERENCE TEMPERATURE     TREF      0.70000E+02   F
PROPERTY TEMP. INCREMENT        DELTAT    0.50000E+02   F
METAL MELTING TEMPERATURE       TMELTM    0.25000E+04   F

$ HIGH TEMPERATURE HIGH STRENGTH NICKEL BASE ALLOY  MAR-M 246
$ SOURCE: INTERNATIONAL NICKEL CO.
MAR-M 246
WEIGHT DENSITY                  RHOM      0.30500E+00   lb/in**3
RHOM EXPONENT                   EXPRH     0.00000E+00   non-dim
ELASTIC MODULUS                 EM        0.29800E+08   psi
EM EXPONENT                     EXPEM     0.23499E+00   non-dim
POISSON"S RATIO                 NUM       0.30000E+00   non-dim
NUM EXPONENT                    EXPNU     0.00000E+00   non-dim
THERMAL EXPANSION COEFF.        ALPHAM    6.15000E-06   in/in/F
ALPHA EXPONENT                  EXPAL     0.35999E+00   non-dim
HEAT CONDUCTIVITY               KM        0.88000E+02   BTU/ft**2/in/hr/F
KM EXPONENT                     EXPKM     0.56999E+00   non-dim
SPECIFIC HEAT                   CM        0.10000E+00   BTU/lb/F
CM EXPONENT                     EXPCM     0.20000E+00   non-dim
ULTIMATE TENSILE STRENGTH       STUM      0.14000E+06   psi
STUM EXPONENT                   EXPST     0.50000E+00   non-dim@
YIELD STRENGTH (0.2% OFFSET)    SYM       0.12500E+06   psi
SYM EXPONENT                    EXPSY     0.31499E+00   non-dim@
REFERENCE RUPTURE STRENGTH      RUP0      0.14000E+06   psi
RUPTURE STRENGTH TEMP. EXPO     RTPEXP    0.38000E+00   non-dim@
RUPTURE STRENGTH TIME  EXPO     RTIEXP    0.15500E+01   non-dim@
TENSILE ELONGATION              EPSTUM    0.50000E+01   %
EPSTUM EXPONENT                 EXPEP     0.50000E+00   non-dim@
METAL REFERENCE TEMPERATURE     TREF      0.70000E+02   F
PROPERTY TEMP. INCREMENT        DELTAT    0.50000E+02   F
METAL MELTING TEMPERATURE       TMELTM    0.24500E+04   F

$ # Indicates that experimental data are not available, exponent is assumed
$ @ Indicates that calibration is required, exponent is assumed
```

APPENDIX B: LISTING OF *propgen.f* MODULE

This is a listing of the FORTRAN module propgen.f which is used to process the material data bank listed in Appendix A. This routine can be used as part of COSMO to generate material properties for the various CSTEM finite element input files or as a stand alone module to generate temperature dependent properties. It requires two input file: material data bank file *matdata.bank* and the T/BEST neutral.file.

```
C-----------------------------------------------------------------
C Routine: PROPGEN.F
C Objective: Generate temperature dependent material properties
C            for use with the T/BEST neutral file.
C            Properties generated are elastic (mechanical and thermal)
C
C Date:     Sep 1995
C
C written by: Galib H. Abumeri
C             NYMA Inc,
C             NASA LeRC group
C             2001 Aerospace PKWY
C             Brook Park, OHIO 44142
C
C Prepared for GE - Aircraft Engine Division - Cincinnati - OH
C Task # 4162, Sep. 1995
C
C-----------------------------------------------------------------
C This routine requires two input files:
C
C 1) neutral.file  (T/BEST standard neutral file)
C 2) matdata.bank  (file containing basic material properties for
C                   titanium and nickel based alloys)
C
C This routine generates one output file: properties.out
C
C.................................
C.. DEFINITION OF KEY PARAMETERS ..
C.................................
C From the T/BEST neutral file, the following parameters are processed:
C
C ARGUMENTS FROM MAIN PROGRAM:
C    TYPE:  ENGINE COMPONENT TYPE (FAN, HPC, LPC, HPT, AND LPT)
C    NCC:   ENGINE COMPONENT NUMBER
C    NIN:   UNIT ON WHICH T/BEST NEUTRAL FILE IS READ
C
C  For each stage of each rotating component: FAN, HPC, LPC, HPT and LPT
C    BLDMAT: Blade material keyword definition, format: 40x, A15
C    DISMAT: Disk  material keyword definition, format: 40x, A15
C    STTIN:  Stage temperature at entrance in Degrees Rankine (40x,e12.5)
C    STTOUT: Stage temperature at outlet in Degrees Rankine (40x,e12.5)
C
C  For non-rotating components, such as burner, inlet, shaft, etc..
C    CMPMAT: Component material keyword definition, format: 40x, A15
C    CTEMP:  Component average temperature in Degrees Rankine: 40x, e12.5
C
C The neutral file parameters listed above are normally extracted and
C stored in the T/BEST neutral file based on the output of the nnepwate
C code which is used for engine cycle and weight analyses. If the temperature
```

```
C of a component was not available, CTEMP is set equal to room temperature.
C
C The module propgen.f will generate temperature dependent material
C properties from reference temperature (70F) up to the maximum temperature
C encountered in a component for an increment defined in the matdata.bank file.
C The properties generated are stored in the file properties.out written on
C unit NPROP=20.
C
C-----------------------------------------------------------------------------
      CHARACTER*3  TYPE
      CHARACTER*15 BLDMAT(20),DISMAT(20),CMPMAT
      CHARACTER*80 TLINE
      DIMENSION TIN(20),TOUT(20)
      NIN   = 10
      NMA   = 15
      NPROP = 20
      OPEN(UNIT=NIN,FILE='neutral.file',STATUS='UNKNOWN')
      OPEN(UNIT=NPROP,FILE='properties.out',STATUS='UNKNOWN')
      OPEN(UNIT=NMA,FILE='matdata.bank',status='UNKNOWN')
      WRITE(NPROP,10005)
      WRITE(NPROP,10010)
      WRITE(NPROP,10015)
      WRITE(NPROP,10020)
100   READ(NIN,'(80A)',END=10000) TLINE
      IF(TLINE(1:22).EQ.'ENGINE COMPONENT TYPE:') THEN
        BACKSPACE NIN
        READ(NIN,'(23X,A3,14X,I5)') TYPE,NCC
        NSTAGE = 0
        ICALL  = 0
        IF(TYPE.EQ.'FAN') CALL CMPN1(TIN,TOUT,BLDMAT,DISMAT,NSTAGE,NIN)
        IF(TYPE.EQ.'HPC') CALL CMPN1(TIN,TOUT,BLDMAT,DISMAT,NSTAGE,NIN)
        IF(TYPE.EQ.'LPC') CALL CMPN1(TIN,TOUT,BLDMAT,DISMAT,NSTAGE,NIN)
        IF(TYPE.EQ.'HPT') CALL CMPN1(TIN,TOUT,BLDMAT,DISMAT,NSTAGE,NIN)
        IF(TYPE.EQ.'LPT') CALL CMPN1(TIN,TOUT,BLDMAT,DISMAT,NSTAGE,NIN)
        IF(TYPE.EQ.'FAN') ICALL = 1
        IF(TYPE.EQ.'HPC'.OR.TYPE.EQ.'LPC') ICALL = 1
        IF(TYPE.EQ.'HPT'.OR.TYPE.EQ.'LPT') ICALL = 1
        IF(TYPE.EQ.'INL') CALL CMPN2(CTEMP,CMPMAT,NIN)
        IF(TYPE.EQ.'DUC') CALL CMPN2(CTEMP,CMPMAT,NIN)
        IF(TYPE.EQ.'PBU') CALL CMPN2(CTEMP,CMPMAT,NIN)
        IF(TYPE.EQ.'SHA') CALL CMPN2(CTEMP,CMPMAT,NIN)
        IF(TYPE.EQ.'FMI') CALL CMPN2(CTEMP,CMPMAT,NIN)
        IF(TYPE.EQ.'NOZ') CALL CMPN2(CTEMP,CMPMAT,NIN)
         IF(ICALL.EQ.1) THEN
           DO I=1,NSTAGE
            IF(TOUT(I).LT.TIN(I))  TUPPER = TIN(I)
            IF(TOUT(I).GT.TIN(I))  TUPPER = TOUT(I)
            IFLAG = 1
            CALL PROPT(TUPPER,BLDMAT,TYPE,I,NCC,NPROP,IFLAG,NMA)
            IFLAG = 2
            CALL PROPT(TUPPER,DISMAT,TYPE,I,NCC,NPROP,IFLAG,NMA)
          END DO
         ENDIF
       IF(ICALL.EQ.0) THEN
         IFLAG = 3
         TUPPER = CTEMP
         CALL PROPT(TUPPER,CMPMAT,TYPE,0,NCC,NPROP,IFLAG,NMA)
       ENDIF
```

```
      ENDIF
      GO TO 100
10000 CONTINUE
C-----------------------------------------------------------------------
C  CALL RUPTURE TO GENERATE TABLES OF t/tf VERSUS RUPTURE STRENGTH FOR
C  MATERIALS IN THE DATA BANK.
C-----------------------------------------------------------------------
      CALL RUPTURE(NMA,NPROP)
      CLOSE(UNIT=NPROP)
      CLOSE(UNIT=NIN)
      CLOSE(UNIT=NMA)
10005 FORMAT('*** T/BEST TEMPERATURE DEPENDENT ',
     +       'MATERIAL PROPERTIES DATA BANK FILE ***',/,
     +       '---------------------------------',
     +       '---------------------------------------')

10010 FORMAT(' MECHANICAL PROPERTIES:',/,
     +       ' ---------------------',/,
     +       ' RHO  = WEIGHT DENSITY (lb/in**3),',/,
     +       ' E    = ELASTIC MODULUS (psi)',/,
     +       ' NU   = POISSON"S RATIO (non-dim),',/,
     +       ' G    = SHEAR MODULUS (psi),')
10015 FORMAT(' STU  = ULTIMATE TENSILE STRENGTH (psi),',/,
     +       ' SY   = 0.2% OFFSET YIELD STRENGTH (psi), '/,
     +       ' EPST = TENSILE ELONGATION (%), ',/)

10020 FORMAT(' THERMAL PROPERTIES:',/,
     +       ' ------------------',/,
     +' TM    = MELTING TEMPERATURE (F), ',/,
     +' TREF  = REFERENCE TEMPERATURE (F),',/,
     +' K     = THERMAL CONDUCTIVITY (BTU-FT**2/hr/in/F), ',/,
     +' ALPHA = THERMAL EXPANSION COEFF. (in/in/F) ',/,
     +' C     = SPECIFIC HEAT (BTU/lb/F) ')
      STOP
      END

C-----------------------------------------------------------------
C SUBROUTINE CMPN1
C WRITTEN BY :       GALIB H. ABUMERI  NYMA / NASA LERC
C DATE:              SEP 1995
C OBJECTIVE: READ NEUTRAL FILE AND PICK UP THE MATERIAL KEYWORD
C            DEFINITION AND TEMPERATURE RANGE FOR BLADES AND DISKS
C
C ARGUMENTS TO MAIN PRTOGRAM:
C    NSTAGE: NUMBER OF STAGES PER COMPONENT
C    BLDMAT: BLADE MATERIAL WHICH IS DEFINED UNDER THE KEYWORD BLDMAT
C            FOR EACH STAGE IN THE NEUTRAL FILE
C    DISMAT: MATERIAL KEYWORD
C    TIN   : TEMPERATURE AT ENTRANCE (F)
C    TOUT  : TEMPERATURE AT OUTLET (F)
C
C-----------------------------------------------------------------
      SUBROUTINE CMPN1(TIN,TOUT,BLDMAT,DISMAT,NSTAGE,NIN)
      CHARACTER*15 BLDMAT(20),DISMAT(20)
      CHARACTER*80 TLINE
      DIMENSION TIN(20),TOUT(20)
 100  READ(NIN,'(80A)',END=10000) TLINE
      IF(TLINE(31:36).EQ.'NSTAGE') GO TO 200
```

```
      GO TO 100
 200  BACKSPACE NIN
      READ(NIN,'(40X,I5)') NSTAGE
      IE = 1
 300  READ(NIN,'(80A)',END=10000) TLINE
      IF(TLINE(31:36).EQ.'STTIN ') THEN
         BACKSPACE NIN
         READ(NIN,'(40X,E12.5)') TIN(IE)
      ENDIF
      IF(TLINE(31:36).EQ.'STTOUT') THEN
         BACKSPACE NIN
         READ(NIN,'(40X,E12.5)') TOUT(IE)
      ENDIF
      IF(TLINE(31:36).EQ.'BLDMAT') THEN
         BACKSPACE NIN
         READ(NIN,'(40X,A15)') BLDMAT(IE)
      ENDIF
      IF(TLINE(31:36).EQ.'DISMAT') THEN
         BACKSPACE NIN
         READ(NIN,'(40X,A15)') DISMAT(IE)
         IF(IE.EQ.NSTAGE) GO TO 10000
         IE = IE + 1
      ENDIF
      GO TO 300
10000 CONTINUE
      RETURN
      END

C---------------------------------------------------------------------
C SUBROUTINE CMPN2
C WRITTEN BY :        GALIB H. ABUMERI  NYMA / NASA LERC
C DATE:               SEP 1995
C OBJECTIVE: READ NEUTRAL FILE AND PICK UP THE MATERIAL KEYWORD
C            DEFINITION AND TEMPERATURE FOR NON-ROTATING ENGINE COMPONENTS
C            SUCH AS BURNER, INLET, AUGMENTER, MIXER, AND SHAFT.
C
C
C ARGUMENTS TO MAIN PRTOGRAM:
C    CMPMAT: COMPONENT MATERIAL KEYWORD DEFINITION
C    CTEMP : AVERAGE COMPONENT TEMPERATURE (F)
C---------------------------------------------------------------------
      SUBROUTINE CMPN2(CTEMP,CMPMAT,NIN)
      CHARACTER*15 CMPMAT
      CHARACTER*80 TLINE
300   READ(NIN,'(80A)',END=10000) TLINE
      IF(TLINE(31:36).EQ.'CMPMAT') THEN
         BACKSPACE NIN
         READ(NIN,'(40X,A15)') CMPMAT
      ENDIF
      IF(TLINE(31:36).EQ.'CTEMP ') THEN
         BACKSPACE NIN
         READ(NIN,'(40X,E12.5)') CTEMP
         GO TO 10000
      ENDIF
      GO TO 300
10000 CONTINUE
      RETURN
```

```
      END
C-------------------------------------------------------------------------
C SUBROUTINE PROPT
C WRITTEN BY : GALIB H. ABUMERI  NYMA / NASA LERC
C DATE: SEP 1995
C*************************************************************************
C OBJECTIVE: GENERATE ELASTIC MATERIAL PROPERTIES FOR EACH BLADE, DISK,
C            AND OTHER COMPONENTS SUCH AS BURNER, DUCT, ETC...
C            PROPERTIES GENERATED ARE TEMPERATURE DEPENDENT AND VARY
C            FROM REFERENCE TEMPERATURE TO THE TUPPER ONE BY INCEREMENT
C            OF DELTAT.
C*************************************************************************
C
C ARGUMENT RECEIVED FROM MAIN PROGRAM:
C    TYPE:  ENGINE COMPONENT TYPE (FAN, HPC, LPC, HPT, AND LPT)
C    NCC:   ENGINE COMPONENT NUMBER
C    NMA:   UNIT ON WHICH MATERIAL DATA BANK FILE (matadata.bank) IS READ
C    TUPPER: UPPER TEMPERATURE LIMIT OF A COMPONENT, PROPERTIES ARE
C            GENERATED FROM AT THE REFERENCE TEMPERATURE TO TUPPER
C
C ARGUMENT SENT BACK TO MAIN PROGRAM:
C    NONE
C ------------------------------------------------------------------------
C GENERATE NOW A TABLE OF TEMPERATURE DEPENDENT ELASTIC MATERIAL
C PROPERTIES BASED ON THE MULTI FACTOR EQUATION:
C
C FOR MECHANICAL PROPERTIES USE : E(T) = E0*[(TM-T)/(TM-T0)]*EXPO
C
C             WHERE E(T) = ELASTIC MODULS AS A FUNCTION OF TEMP.
C                   E0   = EM ELASTIC MODULUS AT ROOM TEMPERATURE
C                   TM   = TMELTM METAL MELTING TEMPERATURE
C                   T0   = REFERENCE TEMPERATURE NORMALLY 70 DEG. F
C                   EXPO = EXPONENT SET TO 0.5 FOR ELASTIC MODULUS,
C                          STRENGTH AND STRAIN AND SET TO -0.5 FOR POISSON'S
C                          RATIO.
C
C  FOR THERMAL PROPERTIES USE: K(T) = K0*[(TM-T0)/(TM-T)]**EXPO
C  THIS IS USED FOR EXPANSION COEFFICIENT, THERMAL CONDUCTIVITY,
C  AND SPECIFIC HEAT. THE EXPONENT IS SET FOR 0.5 FOR NOW.
C
C ------------------------------------------------------------------------
C  Content of matdata.bank for each material; Format (40X,E12.5)
C   (A line beginning with a "$" is used for comments only)
C
C
C     Ti-5621S                     (MATERIAL KEYWORD A15)
C     WEIGHT DENSITY                RHOM       lb/in**3
C     RHOM EXPONENT                 EXPRH      non-dim
C     ELASTIC MODULUS               EM         psi
C     EM EXPONENT                   EXPEM      non-dim
C     POISSON"S RATIO               NUM        non-dim
C     NUM EXPONENT                  EXPNU      non-dim
C     THERMAL EXPANSION COEFF.      ALPHAM     in/in/F
C     ALPHA EXPONENT                EXPAL      non-dim
C     HEAT CONDUCTIVITY             KM         BTU/ft**2/in/hr/F
C     KM EXPONENT                   EXPKM      non-dim
C     SPECIFIC HEAT                 CM         BTU/lb/F
```

```
C      CM EXPONENT                        EXPCM       non-dim
C      ULTIMATE TENSILE STRENGTH          STUM        psi
C      STM EXPONENT                       EXPST       non-dim
C      YIELD STRENGTH (0.2% OFFSET)       SYM         psi
C        SYM EXPONENT                       EXPSY       non-dim
C        REFERENCE RUPTURE STRENGTH         RUP0        psi
C        RUPTURE STRENGTH TEMP. EXPO        RTPEXP      non-dim
C        RUPTURE STRENGTH TIME  EXPO        RTIEXP      non-dim
C      TENSILE ELONGATION                 EPSTM       %
C      EPSTM EXPONENT                     EXPEP       non-dim
C      METAL REFERENCE TEMPERATURE        TREF        F
C      PROPERTY TEMP. INCREMENT           DELTAT      F
C      METAL MELTING TEMPERATURE          TMELTM      F
C
C---------------------------------------------------------------------------
C Definition of generated properties:
C   Mechanical:
C     E     = elastic modulus (psi)
C     G     = shear modulus (psi)
C     RHO   = density (lb/in^3, does not vary with temperature)
C     NU    = poisson's ratio (does not vary with temperature)
C     STU   = ultimate tensile strength (psi)
C     EPST  = tensile elongation
C     SY    = 0.2% offset yield strength
C
C   Thermal:
C     ALPHA= Thermal expansion coeff.  in/in/F
C     K     = Heat conductivity BTU/ft**2/in/hr/F
C     C     = Specific heat BTU/lb/F
C
C---------------------------------------------------------------------------
C IFLAG = 1  FOR BLADE PROPERTIES
C IFLAG = 2  DISK PROPERTIES
C IFLAG = 3  OTHER COMPONENTS
C---------------------------------------------------------------------------
      SUBROUTINE PROPT(TUPPER,MATKEY,TYPE,NS,NCC,NPROP,IFLAG,NMA)
      CHARACTER*3  TYPE
      CHARACTER*5  WHAT,CHECK
      CHARACTER*15 MATKEY
      CHARACTER*80 TLINE
      REAL K(200),KM,NU(200),NUM
      DIMENSION E(200),G(200),STU(200),RHO(200),
     +          C(200),ALPHA(200),EPST(200),T(200),SY(200)
      REWIND NMA
      WRITE(7,'(A15)') MATKEY
      IF(IFLAG.EQ.1) WHAT='BLADE'
      IF(IFLAG.EQ.2) WHAT='DISK '
100   READ(NMA,'(80A)',END=10000) TLINE
      IF(TLINE(1:15).NE.MATKEY) GO TO 100
         READ(NMA,'(40X,E12.5)') RHMO
         READ(NMA,'(40X,E12.5)') EXPRH
         READ(NMA,'(40X,E12.5)') EM
         READ(NMA,'(40X,E12.5)') EXPEM
         READ(NMA,'(40X,E12.5)') NUM
         READ(NMA,'(40X,E12.5)') EXPNU
         READ(NMA,'(40X,E12.5)') ALPHAM
         READ(NMA,'(40X,E12.5)') EXPAL
         READ(NMA,'(40X,E12.5)') KM
```

```
      READ(NMA,'(40X,E12.5)') EXPKM
      READ(NMA,'(40X,E12.5)') CM
      READ(NMA,'(40X,E12.5)') EXPCM
      READ(NMA,'(40X,E12.5)') STUM
      READ(NMA,'(40X,E12.5)') EXPST
      READ(NMA,'(40X,E12.5)') SYM
      READ(NMA,'(40X,E12.5)') EXPSY
      READ(NMA,'(40X,E12.5)') RUP0
      READ(NMA,'(40X,E12.5)') RTPEXP
      READ(NMA,'(40X,E12.5)') RTIEXP
      READ(NMA,'(40X,E12.5)') EPSTM
      READ(NMA,'(40X,E12.5)') EXPEP
      READ(NMA,'(40X,E12.5)') TREF
      READ(NMA,'(40X,E12.5)') DELTAT
      READ(NMA,'(40X,E12.5)') TMELTM

      TM = TMELTM
      T0 = TREF
C------
C  ROUND TUPPER TO NEAREST 0
C------
      WRITE(CHECK,'(F5.0)') TUPPER
      CHECK(4:4) = '0'
      READ(CHECK,'(F5.0)') TLIM
C------
C  DETERMINE THE NUMBER OF VARIOUS TEMPERATURES AT WHICH TEMPERATURE
C  DEPENDENT PROPERTIES ARE GENERATED
C------
      TEMPC = TLIM - TREF
      ICOUNT = INT(TEMPC/DELTAT) + 1
      TINCR = 0.0
      DO I=1,ICOUNT+1
       T(I) = TREF +  TINCR
       IF(I.EQ.(ICOUNT+1))      T(I) = TLIM
C--------
C DETERMINE MECHANICAL PROPERTIES FIRST.
C THE EXPONENTS FOR DENSITY AND POISSON'S RATIO ARE SET TO ZERO
C BECAUSE IT IS ASSUMED THAT THESE PROPERTIES WILL NOT VARY WITH
C TEMPERATURE.
C-------
       RATIOA = TM-T(I)
       IF(RATIOA.LT.0.001)  RATIOA = -1.0*RATIOA
       NU(I)    = NUM *  1.0
       RHO(I)   = RHMO *  1.0
       E(I)     = EM  *   (RATIOA/(TM-T0))**EXPEM
       G(I)     = E(I) / (2.0*(1.0 + NU(I)))
       STU(I)   = STUM *  (RATIOA/(TM-T0))**EXPST
       SY(I)    = SYM *   (RATIOA/(TM-T0))**EXPSY
       EPST(I)  = EPSTM * (RATIOA/(TM-T0))**EXPEP
C---
C DETERMINE THERMAL PROPERTIES
C---
       ALPHA(I) = ALPHAM * ((TM-T0)/RATIOA)**EXPAL
       C(I)     = CM *     ((TM-T0)/RATIOA)**EXPCM
       K(I)     = KM *     ((TM-T0)/RATIOA)**EXPKM
       TINCR = TINCR + DELTAT
      END DO
     ITOTAL = ICOUNT + 1
```

```
       IF(IFLAG.EQ.1.OR.IFLAG.EQ.2) WRITE(NPROP,1010)
     +                              TYPE,WHAT,NCC,NS,MATKEY
       IF(TYPE.EQ.'PBU') WHAT ='RNER '
       IF(TYPE.EQ.'FMI') WHAT ='XER  '
       IF(TYPE.EQ.'DUC') WHAT ='T    '
       IF(TYPE.EQ.'NOZ') WHAT ='ZLE  '
       IF(TYPE.EQ.'AUG') WHAT = 'MENTR'
       IF(TYPE.EQ.'DUC') WHAT ='T    '
       IF(TYPE.EQ.'SHA') WHAT ='FT   '
       IF(TYPE.EQ.'INL') WHAT ='ET   '
       IF(IFLAG.EQ.3) WRITE(NPROP,1015)
     +                              TYPE,WHAT,NCC,MATKEY
       WRITE(NPROP,1016) TREF,TLIM,TMELTM
       IMANY = 7
       ITIME = ITOTAL / IMANY
       IBEG1 = 1
       IBEG2 = IMANY
       WRITE(NPROP,*)
       DO I=1,ITIME
        WRITE(NPROP,1020) (T(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1030) (RHO(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1035) (E(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1040) (NU(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1045) (G(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1050) (STU(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1060) (SY(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1065) (EPST(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1080) (K(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1085) (ALPHA(KK),KK=IBEG1,IBEG2)
        WRITE(NPROP,1090) (C(KK),KK=IBEG1,IBEG2)
        IBEG1 = IBEG2 + 1
        IBEG2 = IBEG2 + IMANY
       END DO
       IREST = ITOTAL - ITIME*IMANY
       IF(IREST.GT.0) IBEG3 = ITIME*IMANY + 1
        IF(IREST.GT.0) THEN
          IR = IBEG3 + IREST - 1
          WRITE(NPROP,1020) (T(KK),KK=IBEG3,IR)
          WRITE(NPROP,1030) (RHO(KK),KK=IBEG3,IR)
          WRITE(NPROP,1035) (E(KK),KK=IBEG3,IR)
          WRITE(NPROP,1040) (NU(KK),KK=IBEG3,IR)
          WRITE(NPROP,1045) (G(KK),KK=IBEG3,IR)
          WRITE(NPROP,1050) (STU(KK),KK=IBEG3,IR)
          WRITE(NPROP,1060) (SY(KK),KK=IBEG3,IR)
          WRITE(NPROP,1065) (EPST(KK),KK=IBEG3,IR)
          WRITE(NPROP,1080) (K(KK),KK=IBEG3,IR)
          WRITE(NPROP,1085) (ALPHA(KK),KK=IBEG3,IR)
          WRITE(NPROP,1090) (C(KK),KK=IBEG3,IR)
        ENDIF
       WRITE(NPROP,1095)
10000 CONTINUE
1010  FORMAT(//,72('-'),/,
     +'COMPONENT TYPE:',A3,1X,A5,', COMPONENT #:',I3,
     +       ', STAGE #:',I3,/,'MATERIAL TYPE:',A15)
1015  FORMAT('COMPONENT TYPE:',A3,A5,', COMPONENT #:',I3,
     +       /,' MATERIAL TYPE:',A15)
1016  FORMAT('REFERENCE TEMP.=',F5.0,' F, MAXIMUM TEMP.=',F5.0,
     +       ' F, MELTING TEMP.=', F5.0,' F')
```

```
1020  FORMAT('     TEMP',7(F8.1,' F'))
1030  FORMAT(' RHO    ',7(E10.3))
1035  FORMAT(' E      ',7(E10.3))
1040  FORMAT(' NU     ',7(E10.3))
1045  FORMAT(' G      ',7(E10.3))
1050  FORMAT(' STU    ',7(E10.3))
1060  FORMAT(' SY     ',7(E10.3))
1065  FORMAT(' EPST   ',7(E10.3))
1080  FORMAT(' K      ',7(E10.3))
1085  FORMAT(' ALPHA  ',7(E10.3))
1090  FORMAT(' C      ',7(E10.3))
1095  FORMAT(72('-'))
      RETURN
      END

C-----------------------------------------------------------------------------
C SUBROUTINE RUPTURE
C WRITTEN BY : GALIB H. ABUMERI  NYMA / NASA LERC
C DATE: NOV 1995
C*****************************************************************************
C OBJECTIVE: GENERATE RUPTURE STRENGTH FOR EACH MATERIAL IN THE DATA BANK
C            FROM TIME t TO TIME tf. DATA ARE WRITTEN AT THE END OF THE FILE
C            properties.out.
C*****************************************************************************
C
C ARGUMENT RECEIVED FROM MAIN PROGRAM:
C NPROP AND NMA (UNITS FOR propertires.out and matdata.bank files).
C
C ARGUMENT SENT BACK TO MAIN PROGRAM:
C    NONE
C-----------------------------------------------------------------------------
C
C CREEP RUPTURE PROPERTIES GENERATED ARE:
C    RUPTURE STRENGTH AT TIME T, ARE PRINTED AT 5 TEMPERATURES:
C    70, 500, 1000, 1500  & 2000 DEG. F.
C
C PARAMETERS USED HERE ARE:
C
C    TF       = final temperature deg. F
C    DELTAT   = Temperature increment at which property is  generated
C    TUPPER   = Upper temperature limit
C    STR      = Load ratio: stress(t) / reference strength
C    RTPEXP   = exponent of temperature term in multi-factor equation
C    RTIEXP   = exponent of time term in multi-factor equation
C
C In this routine, the time required to rupture is determined for each
c temperature:
C  PP0(i) = is the actual rupture strength to reference strength
C           at room temperature.
C  TEMP(I) = temperature varying from room temperature to 2000 F,
C            TEMP(1) = 70 F, TEMP(2) = 200 F, TEMP(5) = 500 F, etc..
C  TTF(i,j)= is the time ratio t/tf corresponding to PP0(i), If the
C            total time is 1000 hr, then the 100 hour rupture strength
C            is determined at t/tf=0.1.
c
C-----------------------------------------------------------------------------
      SUBROUTINE RUPTURE(NMA,NPROP)
      CHARACTER*80 TLINE
```

```
      CHARACTER*15 MATKEY
      dimension pp0(20),TTF(20,20),TEMP(20)
      REAL NUM,KM
      REWIND NMA
      WRITE(NPROP,3000)
010   READ(NMA,'(80A)',END=20000) TLINE
      IF(TLINE(1:6).EQ.'      ') GO TO 10
      if(tline(1:6).eq.'METAL ') go to 10
      IF(TLINE(1:1).EQ.'$')      GO TO 10
      BACKSPACE NMA
      READ(NMA,'(A15)') MATKEY
      WRITE(NPROP,3010) MATKEY
         READ(NMA,'(40X,E12.5)') RHMO
         READ(NMA,'(40X,E12.5)') EXPRH
         READ(NMA,'(40X,E12.5)') EM
         READ(NMA,'(40X,E12.5)') EXPEM
         READ(NMA,'(40X,E12.5)') NUM
         READ(NMA,'(40X,E12.5)') EXPNU
         READ(NMA,'(40X,E12.5)') ALPHAM
         READ(NMA,'(40X,E12.5)') EXPAL
         READ(NMA,'(40X,E12.5)') KM
         READ(NMA,'(40X,E12.5)') EXPKM
         READ(NMA,'(40X,E12.5)') CM
         READ(NMA,'(40X,E12.5)') EXPCM
         READ(NMA,'(40X,E12.5)') STUM
         READ(NMA,'(40X,E12.5)') EXPST
         READ(NMA,'(40X,E12.5)') SYM
         READ(NMA,'(40X,E12.5)') EXPSY
         READ(NMA,'(40X,E12.5)') RUP0
         READ(NMA,'(40X,E12.5)') RTPEXP
         READ(NMA,'(40X,E12.5)') RTIEXP
         READ(NMA,'(40X,E12.5)') EPSTM
         READ(NMA,'(40X,E12.5)') EXPEP
         READ(NMA,'(40X,E12.5)') TREF
         READ(NMA,'(40X,E12.5)') DELTAT
         READ(NMA,'(40X,E12.5)') TMELTM

        T0 = TREF
        TF = 2006.
        DELTAT = 100.
        TUPPER = 2000.
        TLOWER = 70.0
        STR    = 0.9
        P0     = RUP0
        EXPO1  = RTPEXP
        EXPO2  = RTIEXP
        RINC = 0.0
        do i=1,20
          pp0(i) = RINC
          RINC =  RINC + 0.05
        end do
        CCC = (TUPPER-TLOWER)/DELTAT
        ICOUNT  = INT(CCC)
          TPER= TLOWER
          DO 80  N=1,20
           TEMP(N) = TPER
           TTERM =    ((Tf-TEMP(N))/(Tf-T0))**EXPO1
          do 69 m=1,20
```

```
          RATIO1 = PP0(m) / TTERM
          ttime = 0.0
         do 70 La=1,1000
          ttime = ttime + 0.001
          ddd = (1.0 - (STR*ttime))**expo2
          ttf(m,n) = 0.0
          if(abs(DDD-RATIO1).lt.0.001) ttf(m,n) = ttime
          if(abs(DDD-RATIO1).lt.0.001) go to 71
70       continue
71       continue
69       continue
          TPER = TPER + DELTAT
          if(n.eq.1) tper = 200.0
80     CONTINUE
       write(nprop,3015)
       do 300 i=1,20
       write(NPROP,'(F10.0,5f10.3)') P0*PP0(I),TTF(i,1),TTF(i,5),
     +                               TTF(i,10),TTF(i,15),TTF(i,20)
300    continue
       WRITE(NPROP,*)
       WRITE(NPROP,*)
       go to 10
20000  continue
       write(NPROP,3020)
3000  FORMAT(//,72('-'),/,' RUPTURE STRENGTH DATA FOR ALL METALS IN ',
     +                  ' MATERIAL DATA BANK',//)
3010  FORMAT(' MATERIAL: ', A15,/)
3015  format(
     +'  RUPTURE      TIME      TIME      TIME      TIME      TIME',/,
     +'  STRENGTH     t/tf      t/tf      t/tf      t/tf      t/tf',/,
     +'  (psi)    TEMP= 70F     500F      1000F     1500F     2000F',/,
     +'  -----    ---------    ------    ------    ------    ------')
3020  FORMAT(72('-'))
      RETURN
      END
```

APPENDIX C: PARTIAL LISTING OF T/BEST *neutral.file*

The following is a partial listing of the T/BEST neutral file containing various material and temperature keywords that are processed by the property generator *propgen.f* . The properties are generated for each engine component that is available in the neutral file.

```
 ***  T/BEST EXECUTIVE SYSTEM - NEUTRAL FILE  ***
------------------------------------------------
ENGINE COMPONENT TYPE: FAN      NCC            2
NUMBER OF STAGES                NSTAGE         3
MINIMUM CRUISE SPEED            RPMCR       0.54698E+04
ROTOR SPEED                     RPM         0.61585E+04
MAXIMUM ROTOR SPEED             RPMAX       0.68472E+04
BLADE TAPER RATIO (HUB/TIP)     TR          0.55600E+00
UPSTREAM HUB RADIUS             RIUP1       0.11000E+02    (in.)
DOWNSTREAM HUB RADIUS           RIDW1       0.23000E+02    (in.)
UPSTREAM SHROUD RADIUS          ROUP1       0.29000E+02    (in.)
DUCT MACH NUMBER (INLET)        MACHIN      0.60000E+00
FLOW VELOCITY (INLET)           VELIN       0.64700E+03    (ft/sec)
DUCT MACH NUMBER (OUTLET)       MACHOUT     0.50000E+00
FLOW VELOCITY (OUTLET)          VELOUT      0.68200E+03    (ft/sec)
  STAGE NUMBER                  NS             1
   NUMBER OF BLADES             NB            33
   STAGE WEIGHT                 NSTW        0.92600E+03    (lb)
   HUB RADIUS                   RHBA        0.10990E+02    (in)
   TIP RADIUS                   RTBA        0.28930E+02    (in.)
   ASPECT RATIO                 AR          0.30000E+01
   BLADE ROOT ANGLE             THER        0.17969E+02    (deg.)
   STAGE LENGTH                 STL         0.13000E+02    (in.)
   BLADE BROACH ANGLE           BRANG       0.00000E+00    (deg.)
   BLADE STAGGER ANGLE          STAGG       0.35000E+02    (deg.)
   1ST STATION CHORD LENGTH     CHORD(1)    0.42736E+01    (in.)
   STAGE PRESSURE RATIO         PR          0.17411E+01
   STAGE PRESSURE (IN)          STPIN       0.10944E+02    (lb/in^2)
   STAGE TEMPERATURE (IN)       STTIN       0.59310E+02    (F)
   STAGE TEMPERATURE (OUT)      STTOUT      0.15898E+03    (F)
   STAGE MASS FLOW RATE         STAGEF      0.61750E+03    (lb/sec)
   BLADE MATERIAL               BLDMAT     Ti-4Al-4Mn
   AIRFOIL DEFINITION           AIRCODE    NACA 64-206
.............
   DISK MATERIAL                DISMAT     Ti-4Al-4Mn
.............
  STAGE NUMBER                  NS             2
   NUMBER OF BLADES             NB            49
   STAGE WEIGHT                 NSTW        0.49100E+03    (lb)
   HUB RADIUS                   RHBA        0.18540E+02    (in)
   TIP RADIUS                   RTBA        0.28450E+02    (in.)
   ASPECT RATIO                 AR          0.25000E+01
   BLADE ROOT ANGLE             THER        0.17969E+02    (deg.)
   STAGE LENGTH                 STL         0.80000E+01    (in.)
   BLADE BROACH ANGLE           BRANG       0.00000E+00    (deg.)
   BLADE STAGGER ANGLE          STAGG       0.35000E+02    (deg.)
   1ST STATION CHORD LENGTH     CHORD(1)    0.28329E+01    (in.)
   STAGE PRESSURE RATIO         PR          0.16018E+01
   STAGE PRESSURE (IN)          STPIN       0.19055E+02    (lb/in^2)
   STAGE TEMPERATURE (IN)       STTIN       0.15931E+03    (F)
   STAGE TEMPERATURE (OUT)      STTOUT      0.25898E+03    (F)
   STAGE MASS FLOW RATE         STAGEF      0.61750E+03    (lb/sec)
```

```
   BLADE MATERIAL                BLDMAT    Ti-4Al-4Mn
   AIRFOIL DEFINITION            AIRCODE   NACA 64-206 FAN
 ............
   DISK MATERIAL                 DISMAT    Ti-4Al-4Mn
 ............
  STAGE NUMBER                   NS           3
   NUMBER OF BLADES              NB          58
   STAGE WEIGHT                  NSTW       0.38600E+03   (lb)
   HUB RADIUS                    RHBA       0.21350E+02   (in)
   TIP RADIUS                    RTBA       0.27970E+02   (in.)
   ASPECT RATIO                  AR         0.20000E+01
   BLADE ROOT ANGLE              THER       0.17969E+02   (deg.)
   STAGE LENGTH                  STL        0.62000E+01   (in.)
   BLADE BROACH ANGLE            BRANG      0.00000E+00   (deg.)
   BLADE STAGGER ANGLE           STAGG      0.35000E+02   (deg.)
   1ST STATION CHORD LENGTH      CHORD(1)   0.23655E+01   (in.)
   STAGE PRESSURE RATIO          PR         0.15067E+01
   STAGE PRESSURE (IN)           STPIN      0.30523E+02   (lb/in^2)
   STAGE TEMPERATURE (IN)        STTIN      0.25931E+03   (F)
   STAGE TEMPERATURE (OUT)       STTOUT     0.35898E+03   (F)
   STAGE MASS FLOW RATE          STAGEF     0.61750E+03   (lb/sec)
   BLADE MATERIAL                BLDMAT    Ti-4Al-4Mn
............
   DISK MATERIAL                 DISMAT    Ti-4Al-4Mn
............
ENGINE COMPONENT TYPE: HPC       NCC          5
NUMBER OF STAGES                 NSTAGE       5
MINIMUM CRUISE SPEED             RPMCR      0.75352E+04
ROTOR SPEED                      RPM        0.75352E+04
MAXIMUM ROTOR SPEED              RPMAX      0.75352E+04
BLADE TAPER RATIO (HUB/TIP)      TR         0.83300E+00
UPSTREAM HUB RADIUS              RIUP1      0.17000E+02   (in.)
DOWNSTREAM HUB RADIUS            RIDW1      0.20000E+02   (in.)
UPSTREAM SHROUD RADIUS           ROUP1      0.22000E+02   (in.)
DUCT MACH NUMBER (INLET)         MACHIN     0.50000E+00
FLOW VELOCITY (INLET)            VELIN      0.68200E+03   (ft/sec)
DUCT MACH NUMBER (OUTLET)        MACHOUT    0.35000E+00
FLOW VELOCITY (OUTLET)           VELOUT     0.60700E+03   (ft/sec)
  STAGE NUMBER                   NS           1
   NUMBER OF BLADES              NB          45
   STAGE WEIGHT                  NSTW       0.35000E+03   (lb)
   HUB RADIUS                    RHBA       0.16910E+02   (in)
   TIP RADIUS                    RTBA       0.21970E+02   (in.)
   ASPECT RATIO                  AR         0.20000E+01
   BLADE ROOT ANGLE              THER       0.77652E+01   (deg.)
   STAGE LENGTH                  STL        0.53000E+01   (in.)
   BLADE BROACH ANGLE            BRANG      0.00000E+00   (deg.)
   BLADE STAGGER ANGLE           STAGG      0.35000E+02   (deg.)
   1ST STATION CHORD LENGTH      CHORD(1)   0.22995E+01   (in.)
   STAGE PRESSURE RATIO          PR         0.14720E+01
   STAGE PRESSURE (IN)           STPIN      0.48986E+02   (lb/in^2)
   STAGE TEMPERATURE (IN)        STTIN      0.35831E+03   (F)
   STAGE TEMPERATURE (OUT)       STTOUT     0.45811E+03   (F)
   STAGE MASS FLOW RATE          STAGEF     0.49573E+03   (lb/sec)
   BLADE MATERIAL                BLDMAT    ALLOY 713C
   AIRFOIL DEFINITION            AIRCODE   NACA 64-206 FAN
..........
   DISK MATERIAL                 DISMAT    ALLOY 713LC
```

```
.............
 STAGE NUMBER                NS          2
  NUMBER OF BLADES           NB         58
  STAGE WEIGHT               NSTW      0.25700E+03   (lb)
  HUB RADIUS                 RHBA      0.18280E+02   (in)
  TIP RADIUS                 RTBA      0.21970E+02   (in.)
  ASPECT RATIO               AR        0.18500E+01
  BLADE ROOT ANGLE           THER      0.77652E+01   (deg.)
  STAGE LENGTH               STL       0.41000E+01   (in.)
  BLADE BROACH ANGLE         BRANG     0.00000E+00   (deg.)
  BLADE STAGGER ANGLE        STAGG     0.35000E+02   (deg.)
  1ST STATION CHORD LENGTH   CHORD(1)  0.18129E+01   (in.)
  STAGE PRESSURE RATIO       PR        0.14135E+01
  STAGE PRESSURE (IN)        STPIN     0.72108E+02   (lb/in^2)
  STAGE TEMPERATURE (IN)     STTIN     0.46031E+03   (F)
  STAGE TEMPERATURE (OUT)    STTOUT    0.56011E+03   (F)
  STAGE MASS FLOW RATE       STAGEF    0.49573E+03   (lb/sec)
  BLADE MATERIAL             BLDMAT    ALLOY 713C
  AIRFOIL DEFINITION         AIRCODE   NACA 64-206 FAN
.............
  DISK MATERIAL              DISMAT    ALLOY 713LC
............
 STAGE NUMBER                NS          3
  NUMBER OF BLADES           NB         69
  STAGE WEIGHT               NSTW      0.21900E+03   (lb)
  HUB RADIUS                 RHBA      0.19130E+02   (in)
  TIP RADIUS                 RTBA      0.21970E+02   (in.)
  ASPECT RATIO               AR        0.17000E+01
  BLADE ROOT ANGLE           THER      0.77652E+01   (deg.)
  STAGE LENGTH               STL       0.34000E+01   (in.)
  BLADE BROACH ANGLE         BRANG     0.00000E+00   (deg.)
  BLADE STAGGER ANGLE        STAGG     0.35000E+02   (deg.)
  1ST STATION CHORD LENGTH   CHORD(1)  0.15184E+01   (in.)
  STAGE PRESSURE RATIO       PR        0.13684E+01
  STAGE PRESSURE (IN)        STPIN     0.10192E+03   (lb/in^2)
  STAGE TEMPERATURE (IN)     STTIN     0.56131E+03   (F)
  STAGE TEMPERATURE (OUT)    STTOUT    0.66111E+03   (F)
  STAGE MASS FLOW RATE       STAGEF    0.49573E+03   (lb/sec)
  BLADE MATERIAL             BLDMAT    ALLOY 713C
  AIRFOIL DEFINITION         AIRCODE   NACA 64-206 FAN
............
  DISK MATERIAL              DISMAT    ALLOY 713LC
............
 STAGE NUMBER                NS          4
  NUMBER OF BLADES           NB         79
  STAGE WEIGHT               NSTW      0.19400E+03   (lb)
  HUB RADIUS                 RHBA      0.19700E+02   (in)
  TIP RADIUS                 RTBA      0.21970E+02   (in.)
  ASPECT RATIO               AR        0.15500E+01
  BLADE ROOT ANGLE           THER      0.77652E+01   (deg.)
  STAGE LENGTH               STL       0.30000E+01   (in.)
  BLADE BROACH ANGLE         BRANG     0.00000E+00   (deg.)
  BLADE STAGGER ANGLE        STAGG     0.35000E+02   (deg.)
  1ST STATION CHORD LENGTH   CHORD(1)  0.13311E+01   (in.)
  STAGE PRESSURE RATIO       PR        0.13325E+01
  STAGE PRESSURE (IN)        STPIN     0.13947E+03   (lb/in^2)
  STAGE TEMPERATURE (IN)     STTIN     0.66131E+03   (F)
  STAGE TEMPERATURE (OUT)    STTOUT    0.76111E+03   (F)
```

STAGE MASS FLOW RATE	STAGEF	0.49573E+03	(lb/sec)
BLADE MATERIAL	BLDMAT	ALLOY 713C	
AIRFOIL DEFINITION	AIRCODE	NACA 64-206 FAN	
....			
DISK MATERIAL	DISMAT	ALLOY 713LC	
....			
STAGE NUMBER	NS	5	
NUMBER OF BLADES	NB	86	
STAGE WEIGHT	NSTW	0.17900E+03	(lb)
HUB RADIUS	RHBA	0.20090E+02	(in)
TIP RADIUS	RTBA	0.21970E+02	(in.)
ASPECT RATIO	AR	0.14000E+01	
BLADE ROOT ANGLE	THER	0.77652E+01	(deg.)
STAGE LENGTH	STL	0.27000E+01	(in.)
BLADE BROACH ANGLE	BRANG	0.00000E+00	(deg.)
BLADE STAGGER ANGLE	STAGG	0.35000E+02	(deg.)
1ST STATION CHORD LENGTH	CHORD(1)	0.12205E+01	(in.)
STAGE PRESSURE RATIO	PR	0.13032E+01	
STAGE PRESSURE (IN)	STPIN	0.18585E+03	(lb/in^2)
STAGE TEMPERATURE (IN)	STTIN	0.76031E+03	(F)
STAGE TEMPERATURE (OUT)	STTOUT	0.86011E+03	(F)
STAGE MASS FLOW RATE	STAGEF	0.49573E+03	(lb/sec)
BLADE MATERIAL	BLDMAT	ALLOY 713C	
AIRFOIL DEFINITION	AIRCODE	NACA 64-206 FAN	
....			
DISK MATERIAL	DISMAT	ALLOY 713LC	
....			
ENGINE COMPONENT TYPE: HPT	NCC	8	
NUMBER OF STAGES	NSTAGE	1	
MINIMUM CRUISE SPEED	RPMCR	0.66700E+04	
ROTOR SPEED	RPM	0.75352E+04	
MAXIMUM ROTOR SPEED	RPMAX	0.84004E+04	
BLADE TAPER RATIO (HUB/TIP)	TR	0.10000E+01	
UPSTREAM HUB RADIUS	RIUP1	0.20000E+02	(in.)
DOWNSTREAM HUB RADIUS	RIDW1	0.20000E+02	(in.)
UPSTREAM SHROUD RADIUS	ROUP1	0.23000E+02	(in.)
DUCT MACH NUMBER (INLET)	MACHIN	0.25000E+00	
FLOW VELOCITY (INLET)	VELIN	0.62800E+03	(ft/sec)
DUCT MACH NUMBER (OUTLET)	MACHOUT	0.45000E+00	
FLOW VELOCITY (OUTLET)	VELOUT	0.10140E+04	(ft/sec)
STAGE NUMBER	NS	1	
NUMBER OF BLADES	NB	98	
STAGE WEIGHT	NSTW	0.96420E+03	(lb)
HUB RADIUS	RHBA	0.19510E+02	(in)
TIP RADIUS	RTBA	0.22550E+02	(in.)
ASPECT RATIO	AR	0.20000E+01	
BLADE ROOT ANGLE	THER	0.00000E+00	(deg.)
STAGE LENGTH	STL	0.35500E+01	(in.)
BLADE BROACH ANGLE	BRANG	0.00000E+00	(deg.)
BLADE STAGGER ANGLE	STAGG	0.35000E+02	(deg.)
1ST STATION CHORD LENGTH	CHORD(1)	0.15200E+01	(in.)
STAGE PRESSURE RATIO	PR	0.23625E+01	
STAGE PRESSURE (IN)	STPIN	0.25886E+03	(lb/in^2)
STAGE TEMPERATURE (IN)	STTIN	0.24143E+04	(F)
STAGE TEMPERATURE (OUT)	STTOUT	0.18843E+04	(F)
STAGE MASS FLOW RATE	STAGEF	0.42849E+03	(lb/sec)
BLADE MATERIAL	BLDMAT	MAR-M 200	
AIRFOIL DEFINITION	AIRCODE	NACA 64-206 TURBINE	

. . . .

DISK MATERIAL	DISMAT	MAR-M 246	

. . . .

NGINE COMPONENT TYPE: LPT	NCC	9	
NUMBER OF STAGES	NSTAGE	2	
MINIMUM CRUISE SPEED	RPMCR	0.54698E+04	
ROTOR SPEED	RPM	0.61585E+04	
MAXIMUM ROTOR SPEED	RPMAX	0.68472E+04	
BLADE TAPER RATIO (HUB/TIP)	TR	0.10000E+01	
UPSTREAM HUB RADIUS	RIUP1	0.19000E+02	(in.)
DOWNSTREAM HUB RADIUS	RIDW1	0.18000E+02	(in.)
UPSTREAM SHROUD RADIUS	ROUP1	0.24000E+02	(in.)
DUCT MACH NUMBER (INLET)	MACHIN	0.43000E+00	
FLOW VELOCITY (INLET)	VELIN	0.96100E+03	(ft/sec)
DUCT MACH NUMBER (OUTLET)	MACHOUT	0.52000E+00	
FLOW VELOCITY (OUTLET)	VELOUT	0.10710E+04	(ft/sec)
STAGE NUMBER	NS	1	
NUMBER OF BLADES	NB	89	
STAGE WEIGHT	NSTW	0.61700E+03	(lb)
HUB RADIUS	RHBA	0.19270E+02	(in)
TIP RADIUS	RTBA	0.23640E+02	(in.)
ASPECT RATIO	AR	0.25000E+01	
BLADE ROOT ANGLE	THER	-0.71250E+01	(deg.)
STAGE LENGTH	STL	0.20400E+01	(in.)
BLADE BROACH ANGLE	BRANG	0.00000E+00	(deg.)
BLADE STAGGER ANGLE	STAGG	0.35000E+02	(deg.)
1ST STATION CHORD LENGTH	CHORD(1)	0.17480E+01	(in.)
STAGE PRESSURE RATIO	PR	0.14002E+01	
STAGE PRESSURE (IN)	STPIN	0.10123E+03	(lb/in^2)
STAGE TEMPERATURE (IN)	STTIN	0.18343E+04	(F)
STAGE TEMPERATURE (OUT)	STTOUT	0.16683E+04	(F)
STAGE MASS FLOW RATE	STAGEF	0.48124E+03	(lb/sec)
BLADE MATERIAL	BLDMAT	MAR-M 200	
AIRFOIL DEFINITION	AIRCODE	NACA 64-206 TURBINE	

. . . .

DISK MATERIAL	DISMAT	MAR-M 246	

. . . .

DISK WEIGHT	DWGHT	0.00000E+00	(lbs)
STAGE NUMBER	NS	2	
NUMBER OF BLADES	NB	101	
STAGE WEIGHT	NSTW	0.95821E+03	(lb)
HUB RADIUS	RHBA	0.18660E+02	(in)
TIP RADIUS	RTBA	0.24130E+02	(in.)
ASPECT RATIO	AR	0.35000E+01	
BLADE ROOT ANGLE	THER	-0.71250E+01	(deg.)
STAGE LENGTH	STL	0.41900E+01	(in.)
BLADE BROACH ANGLE	BRANG	0.00000E+00	(deg.)
BLADE STAGGER ANGLE	STAGG	0.35000E+02	(deg.)
1ST STATION CHORD LENGTH	CHORD(1)	0.15629E+01	(in.)
STAGE PRESSURE RATIO	PR	0.14372E+01	
STAGE PRESSURE (IN)	STPIN	0.10123E+03	(lb/in^2)
STAGE TEMPERATURE (IN)	STTIN	0.16683E+04	(F)
STAGE TEMPERATURE (OUT)	STTOUT	0.13363E+04	(F)
STAGE MASS FLOW RATE	STAGEF	0.48124E+03	(lb/sec)
BLADE MATERIAL	BLDMAT	MAR-M 200	
AIRFOIL DEFINITION	AIRCODE	NACA 64-206 TURBINE	

. . . .

DISK MATERIAL	DISMAT	MAR-M 246	

```
.............
ENGINE COMPONENT TYPE: DUCT    NCC          4
   MATERIAL                    CMPMAT     ALLOY 713LC
   PROCESS                     TYPROC     MAURER
   DUCT WEIGHT                 WGHT        0.28089E+02    (lbs)
   STOCK MATERIAL WEIGHT       SWGHT       0.13044E+02    (lbs)
   MAURER WEIGHT FACTOR        MAURER      0.00000E+00    (lbs)
   COST TO MANUFACTURE ONE     COST1       0.00000E+00    ($)
   INNER RADIUS                RIN         0.22880E+02    (in.)
   OUTER RADIUS                ROUT        0.26390E+02    (in.)
   COMPONENT LENGTH            LENGTH      0.87800E+01    (in.)
   COMMPONENT THICKNESS        CTHK        0.61500E-01    (in.)
   COMPONENT TEMPERATURE       CTEMP       0.35861E+03    (F)
   COMPONENT PRESSURE          CPRESS      0.58637E+02    (lb/in^2)
   MAXIMUM EFFECTIVE STRESS    STRESS      0.00000E+00    (psi)
   MAXIMUM DISPLACEMENT        DISP        0.00000E+00    (in.)
   NATURAL FREQUENCY           FREQ        0.00000E+00    (cps) MODE 1
   CRITICAL BUCKLING LOAD      BUCKLE      0.00000E+00    (lbs)
ENGINE COMPONENT TYPE: DUCT    NCC          6
   MATERIAL                    CMPMAT     ALLOY 713LC
   PROCESS                     TYPROC     MAURER
   DUCT WEIGHT                 WGHT        0.10170E+03    (lbs)
   STOCK MATERIAL WEIGHT       SWGHT       0.49164E+02    (lbs)
   MAURER WEIGHT FACTOR        MAURER      0.00000E+00    (lbs)
   COST TO MANUFACTURE ONE     COST1       0.00000E+00    ($)
   INNER RADIUS                RIN         0.20470E+02    (in.)
   OUTER RADIUS                ROUT        0.21870E+02    (in.)
   COMPONENT LENGTH            LENGTH      0.80700E+01    (in.)
   COMMPONENT THICKNESS        CTHK        0.16560E+00    (in.)
   COMPONENT TEMPERATURE       CTEMP       0.85751E+03    (F)
   COMPONENT PRESSURE          CPRESS      0.28665E+03    (lb/in^2)
   MAXIMUM EFFECTIVE STRESS    STRESS      0.00000E+00    (psi)
   MAXIMUM DISPLACEMENT        DISP        0.00000E+00    (in.)
   NATURAL FREQUENCY           FREQ        0.00000E+00    (cps) MODE 1
   CRITICAL BUCKLING LOAD      BUCKLE      0.00000E+00    (lbs)
ENGINE COMPONENT TYPE: PBUR    NCC          7
   MATERIAL                    CMPMAT     MAR-M 246
   PROCESS                     TYPROC     MAURER
   PBUR WEIGHT                 WGHT        0.84500E+03    (lbs)
   STOCK MATERIAL WEIGHT       SWGHT       0.00000E+00    (lbs)
   MAURER WEIGHT FACTOR        MAURER      0.00000E+00    (lbs)
   COST TO MANUFACTURE ONE     COST1       0.00000E+00    ($)
   INNER RADIUS                RIN         0.19300E+02    (in.)
   OUTER RADIUS                ROUT        0.22885E+02    (in.)
   COMPONENT LENGTH            LENGTH      0.18000E+02    (in.)
   NUMBER OF NOZZLES           NCNOZZ       4
   COMMPONENT THICKNESS        CTHK        0.14600E+00    (in.)
   COMPONENT TEMPERATURE       CTEMP       0.11998E+04    (F)
   COMPONENT PRESSURE          CPRESS      0.35340E+03    (lb/in^2)
   MAXIMUM EFFECTIVE STRESS    STRESS      0.00000E+00    (psi)
   MAXIMUM DISPLACEMENT        DISP        0.00000E+00    (in.)
   NATURAL FREQUENCY           FREQ        0.00000E+00    (cps) MODE 1
   CRITICAL BUCKLING LOAD      BUCKLE      0.00000E+00    (lbs)
ENGINE COMPONENT TYPE: FMIX    NCC          11
   MATERIAL                    CMPMAT     MAR-M 246
   PROCESS                     TYPROC     MAURER
   FMIX WEIGHT                 WGHT        0.10951E+03    (lbs)
   STOCK MATERIAL WEIGHT       SWGHT       0.00000E+00    (lbs)
```

MAURER WEIGHT FACTOR	MAURER	0.00000E+00	(lbs)
COST TO MANUFACTURE ONE	COST1	0.00000E+00	($)
INNER RADIUS	RIN	0.00000E+00	(in.)
OUTER RADIUS	ROUT	0.00000E+00	(in.)
COMPONENT LENGTH	LENGTH	0.16590E+02	(in.)
COMMPONENT THICKNESS	CTHK	0.00000E+00	(in.)
COMPONENT TEMPERATURE	CTEMP	0.70000E+02	(F)
COMPONENT PRESSURE	CPRESS	0.00000E+00	(lb/in^2)
ENGINE COMPONENT TYPE: AUG	NCC	12	
MATERIAL	CMPMAT	ALLOY 713C	
PROCESS	TYPROC	MAURER	
AUG WEIGHT	WGHT	0.74000E+02	(lbs)
STOCK MATERIAL WEIGHT	SWGHT	0.00000E+00	(lbs)
MAURER WEIGHT FACTOR	MAURER	0.00000E+00	(lbs)
COST TO MANUFACTURE ONE	COST1	0.00000E+00	($)
INNER RADIUS	RIN	0.00000E+00	(in.)
OUTER RADIUS	ROUT	0.24779E+02	(in.)
COMPONENT LENGTH	LENGTH	0.00000E+00	(in.)
COMMPONENT THICKNESS	CTHK	0.80000E-01	(in.)
COMPONENT TEMPERATURE	CTEMP	0.19842E+04	(F)
COMPONENT PRESSURE	CPRESS	0.69000E+02	(lb/in^2)
ENGINE COMPONENT TYPE: NOZ	NCC	13	
MATERIAL	CMPMAT	MAR-M 200	
PROCESS	TYPROC	MAURER	
NOZ WEIGHT	WGHT	0.48829E+04	(lbs)
STOCK MATERIAL WEIGHT	SWGHT	0.00000E+00	(lbs)
MAURER WEIGHT FACTOR	MAURER	0.00000E+00	(lbs)
COST TO MANUFACTURE ONE	COST1	0.00000E+00	($)
INNER RADIUS	RIN	0.00000E+00	(in.)
OUTER RADIUS	ROUT	0.00000E+00	(in.)
COMPONENT LENGTH	LENGTH	0.22549E+03	(in.)
COMMPONENT THICKNESS	CTHK	0.10000E+00	(in.)
COMPONENT TEMPERATURE	CTEMP	0.15245E+04	(F)
COMPONENT PRESSURE	CPRESS	0.69000E+02	(lb/in^2)
ENGINE COMPONENT TYPE: SHAF	NCC	17	
MATERIAL	CMPMAT	IN-100	
PROCESS	TYPROC	MAURER	
SHAF WEIGHT	WGHT	0.17877E+03	(lbs)
STOCK MATERIAL WEIGHT	SWGHT	0.00000E+00	(lbs)
MAURER WEIGHT FACTOR	MAURER	0.00000E+00	(lbs)
COST TO MANUFACTURE ONE	COST1	0.00000E+00	($)
INNER RADIUS	RIN	0.23700E+01	(in.)
OUTER RADIUS	ROUT	0.27400E+01	(in.)
COMPONENT LENGTH	LENGTH	0.68400E+02	(in.)
COMMPONENT THICKNESS	CTHK	0.74000E+00	(in.)
COMPONENT TEMPERATURE	CTEMP	0.70000E+02	(F)
COMPONENT PRESSURE	CPRESS	0.00000E+00	(lb/in^2)
SHAFT DN-NUMBER OF BEARINGSDN		0.95000E+00	
ENGINE COMPONENT TYPE: SHAF	NCC	16	
MATERIAL	CMPMAT	IN-100	
PROCESS	TYPROC	MAURER	
SHAF WEIGHT	WGHT	0.11797E+03	(lbs)
STOCK MATERIAL WEIGHT	SWGHT	0.00000E+00	(lbs)
MAURER WEIGHT FACTOR	MAURER	0.00000E+00	(lbs)
COST TO MANUFACTURE ONE	COST1	0.00000E+00	($)
INNER RADIUS	RIN	0.29400E+01	(in.)
OUTER RADIUS	ROUT	0.32100E+01	(in.)
COMPONENT LENGTH	LENGTH	0.29620E+02	(in.)

```
   COMMPONENT THICKNESS        CTHK        0.54000E+00    (in.)
   COMPONENT TEMPERATURE       CTEMP       0.70000E+02    (F)
   COMPONENT PRESSURE          CPRESS      0.00000E+00    (lb/in^2)
   SHAFT DN-NUMBER OF BEARINGSDN           0.13700E+01
ENGINE COMPONENT TYPE: INLT    NCC            1
   MATERIAL                    CMPMAT     B-1900
   PROCESS                     TYPROC     MAURER
   INLT WEIGHT                 WGHT        0.30170E+04    (lbs)
   STOCK MATERIAL WEIGHT       SWGHT       0.00000E+00    (lbs)
   MAURER WEIGHT FACTOR        MAURER      0.00000E+00    (lbs)
   COST TO MANUFACTURE ONE     COST1       0.00000E+00    ($)
   INNER RADIUS                RIN         0.00000E+00    (in.)
   OUTER RADIUS                ROUT        0.00000E+00    (in.)
   COMPONENT LENGTH            LENGTH      0.11284E+03    (in.)
   COMMPONENT THICKNESS        CTHK        0.10000E+00    (in.)
   COMPONENT TEMPERATURE       CTEMP       0.70000E+02    (F)
   COMPONENT PRESSURE          CPRESS      0.00000E+00    (lb/in^2)
etc....
```

APPENDIX D: PARTIAL LISTING OF *properties.out* FILE

In this appendix, a section of *properties.out* file is listed. Note that this file is automatically created while executing the property generator module *propgen.f* . It contains properties from reference to the maximum operating temperature with an increment of DELTAT as defined in the material data bank. The properties are computed and generated for each stage of fan, compressor, turbine, and inlet, burner, mixer, augmenter, shaft, and nozzle. Since *properties.out* is a large file, a limited section is listed here for demonstration purposes only.

```
*** T/BEST TEMPERATURE DEPENDENT MATERIAL PROPERTIES DATA BANK FILE ***
----------------------------------------------------------------------
 MECHANICAL PROPERTIES:
 ---------------------
 RHO  = WEIGHT DENSITY (lb/in**3),
 E    = ELASTIC MODULUS (psi)
 NU   = POISSON"S RATIO (non-dim),
 G    = SHEAR MODULUS (psi),
 STU  = ULTIMATE TENSILE STRENGTH (psi),
 SY   = 0.2% OFFSET YIELD STRENGTH (psi),
 EPST = TENSILE ELONGATION (%),

 THERMAL PROPERTIES:
 ------------------
 TM    = MELTING TEMPERATURE (F),
 TREF  = REFERENCE TEMPERATURE (F),
 K     = THERMAL CONDUCTIVITY (BTU-FT**2/hr/in/F),
 ALPHA = THERMAL EXPANSION COEFF. (in/in/F)
 C     = SPECIFIC HEAT (BTU/lb/F)

..........

----------------------------------------------------------------------
COMPONENT TYPE:HPC BLADE, COMPONENT #:  5, STAGE #:  2
MATERIAL TYPE:ALLOY 713C
REFERENCE TEMP.=  70. F, MAXIMUM TEMP.= 560. F, MELTING TEMP.=2350. F
```

TEMP	70.0 F	120.0 F	170.0 F	220.0 F	270.0 F	320.0 F	370.0 F
RHO	0.286E+00	0.286E+00	0.286E+00	0.286E+00	0.286E+00	0.286E+00	0.286E+00
E	0.299E+08	0.297E+08	0.296E+08	0.294E+08	0.292E+08	0.291E+08	0.289E+08
NU	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00
G	0.115E+08	0.114E+08	0.114E+08	0.113E+08	0.112E+08	0.112E+08	0.111E+08
STU	0.123E+06	0.122E+06	0.120E+06	0.119E+06	0.117E+06	0.116E+06	0.115E+06
SY	0.107E+06	0.106E+06	0.106E+06	0.105E+06	0.104E+06	0.103E+06	0.103E+06
EPST	0.800E+01	0.791E+01	0.782E+01	0.773E+01	0.764E+01	0.755E+01	0.746E+01
K	0.702E+02	0.711E+02	0.720E+02	0.730E+02	0.740E+02	0.750E+02	0.761E+02
ALPHA	0.545E-05	0.549E-05	0.553E-05	0.558E-05	0.562E-05	0.567E-05	0.572E-05
C	0.100E+00	0.101E+00	0.101E+00	0.102E+00	0.103E+00	0.104E+00	0.105E+00

TEMP	420.0 F	470.0 F	520.0 F	560.0 F
RHO	0.286E+00	0.286E+00	0.286E+00	0.286E+00
E	0.287E+08	0.285E+08	0.283E+08	0.282E+08
NU	0.300E+00	0.300E+00	0.300E+00	0.300E+00
G	0.110E+08	0.110E+08	0.109E+08	0.108E+08
STU	0.113E+06	0.112E+06	0.110E+06	0.109E+06
SY	0.102E+06	0.101E+06	0.100E+06	0.995E+05
EPST	0.736E+01	0.726E+01	0.717E+01	0.709E+01
K	0.773E+02	0.785E+02	0.797E+02	0.807E+02
ALPHA	0.577E-05	0.583E-05	0.588E-05	0.593E-05

C	0.106E+00	0.107E+00	0.108E+00	0.108E+00

COMPONENT TYPE:HPC DISK , COMPONENT #: 5, STAGE #: 2
MATERIAL TYPE:ALLOY 713LC
REFERENCE TEMP.= 70. F, MAXIMUM TEMP.= 560. F, MELTING TEMP.=2410. F

TEMP	70.0 F	120.0 F	170.0 F	220.0 F	270.0 F	320.0 F	370.0 F
RHO	0.289E+00	0.289E+00	0.289E+00	0.289E+00	0.289E+00	0.289E+00	0.289E+00
E	0.286E+08	0.284E+08	0.283E+08	0.281E+08	0.279E+08	0.277E+08	0.276E+08
NU	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00
G	0.110E+08	0.109E+08	0.109E+08	0.108E+08	0.107E+08	0.107E+08	0.106E+08
STU	0.130E+06	0.129E+06	0.127E+06	0.126E+06	0.124E+06	0.123E+06	0.121E+06
SY	0.109E+06	0.108E+06	0.107E+06	0.107E+06	0.106E+06	0.105E+06	0.104E+06
EPST	0.150E+02	0.148E+02	0.147E+02	0.145E+02	0.143E+02	0.142E+02	0.140E+02
K	0.675E+02	0.684E+02	0.694E+02	0.704E+02	0.714E+02	0.724E+02	0.735E+02
ALPHA	0.469E-05	0.475E-05	0.481E-05	0.487E-05	0.494E-05	0.501E-05	0.508E-05
C	0.105E+00	0.106E+00	0.106E+00	0.107E+00	0.108E+00	0.109E+00	0.110E+00

TEMP	420.0 F	470.0 F	520.0 F	560.0 F
RHO	0.289E+00	0.289E+00	0.289E+00	0.289E+00
E	0.274E+08	0.272E+08	0.270E+08	0.268E+08
NU	0.300E+00	0.300E+00	0.300E+00	0.300E+00
G	0.105E+08	0.105E+08	0.104E+08	0.103E+08
STU	0.120E+06	0.118E+06	0.117E+06	0.116E+06
SY	0.103E+06	0.102E+06	0.101E+06	0.101E+06
EPST	0.138E+02	0.137E+02	0.135E+02	0.133E+02
K	0.747E+02	0.759E+02	0.771E+02	0.782E+02
ALPHA	0.515E-05	0.523E-05	0.531E-05	0.537E-05
C	0.111E+00	0.111E+00	0.112E+00	0.113E+00

....

COMPONENT TYPE:PBURNER , COMPONENT #: 7
MATERIAL TYPE:MAR-M 246
REFERENCE TEMP.= 70. F, MAXIMUM TEMP.=1200. F, MELTING TEMP.=2450. F

TEMP	70.0 F	120.0 F	170.0 F	220.0 F	270.0 F	320.0 F	370.0 F
RHO	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00
E	0.298E+08	0.297E+08	0.295E+08	0.293E+08	0.292E+08	0.290E+08	0.289E+08
NU	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00
G	0.115E+08	0.114E+08	0.113E+08	0.113E+08	0.112E+08	0.112E+08	0.111E+08
STU	0.140E+06	0.139E+06	0.137E+06	0.136E+06	0.134E+06	0.132E+06	0.131E+06
SY	0.125E+06	0.124E+06	0.123E+06	0.122E+06	0.122E+06	0.121E+06	0.120E+06
EPST	0.500E+01	0.495E+01	0.489E+01	0.484E+01	0.479E+01	0.473E+01	0.467E+01
K	0.880E+02	0.891E+02	0.902E+02	0.913E+02	0.925E+02	0.937E+02	0.950E+02
ALPHA	0.615E-05	0.620E-05	0.625E-05	0.630E-05	0.635E-05	0.640E-05	0.646E-05
C	0.100E+00	0.100E+00	0.101E+00	0.101E+00	0.102E+00	0.102E+00	0.103E+00

TEMP	420.0 F	470.0 F	520.0 F	570.0 F	620.0 F	670.0 F	720.0 F
RHO	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00
E	0.287E+08	0.285E+08	0.284E+08	0.282E+08	0.280E+08	0.278E+08	0.276E+08
NU	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00
G	0.110E+08	0.110E+08	0.109E+08	0.108E+08	0.108E+08	0.107E+08	0.106E+08
STU	0.129E+06	0.128E+06	0.126E+06	0.124E+06	0.123E+06	0.121E+06	0.119E+06
SY	0.119E+06	0.118E+06	0.117E+06	0.116E+06	0.115E+06	0.114E+06	0.113E+06

EPST	0.462E+01	0.456E+01	0.450E+01	0.444E+01	0.438E+01	0.432E+01	0.426E+01
K	0.964E+02	0.977E+02	0.992E+02	0.101E+03	0.102E+03	0.104E+03	0.106E+03
ALPHA	0.651E-05	0.657E-05	0.663E-05	0.669E-05	0.676E-05	0.683E-05	0.690E-05
C	0.103E+00	0.104E+00	0.104E+00	0.105E+00	0.105E+00	0.106E+00	0.107E+00
TEMP	770.0 F	820.0 F	870.0 F	920.0 F	970.0 F	1020.0 F	1070.0 F
RHO	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00	0.305E+00
E	0.275E+08	0.273E+08	0.271E+08	0.269E+08	0.267E+08	0.264E+08	0.262E+08
NU	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00	0.300E+00
G	0.106E+08	0.105E+08	0.104E+08	0.103E+08	0.103E+08	0.102E+08	0.101E+08
STU	0.118E+06	0.116E+06	0.114E+06	0.112E+06	0.110E+06	0.109E+06	0.107E+06
SY	0.112E+06	0.111E+06	0.110E+06	0.109E+06	0.108E+06	0.106E+06	0.105E+06
EPST	0.420E+01	0.414E+01	0.407E+01	0.401E+01	0.394E+01	0.388E+01	0.381E+01
K	0.107E+03	0.109E+03	0.111E+03	0.113E+03	0.115E+03	0.118E+03	0.120E+03
ALPHA	0.697E-05	0.705E-05	0.713E-05	0.721E-05	0.730E-05	0.739E-05	0.748E-05
C	0.107E+00	0.108E+00	0.109E+00	0.109E+00	0.110E+00	0.111E+00	0.112E+00

....

....

RUPTURE STRENGTH DATA FOR ALL METALS IN MATERIAL DATA BANK

....

MATERIAL: B-1900

RUPTURE STRENGTH (psi)	TIME t/tf TEMP= 70F	TIME t/tf 500F	TIME t/tf 1000F	TIME t/tf 1500F	TIME t/tf 2000F
0.	0.000	0.000	0.000	0.000	0.000
7050.	0.949	0.939	0.921	0.886	0.448
14100.	0.858	0.843	0.815	0.761	0.074
21150.	0.783	0.763	0.727	0.656	0.000
28200.	0.717	0.692	0.649	0.564	0.000
35250.	0.656	0.627	0.577	0.479	0.000
42300.	0.600	0.567	0.511	0.401	0.000
49350.	0.546	0.510	0.448	0.326	0.000
56400.	0.495	0.456	0.388	0.256	0.000
63450.	0.447	0.405	0.331	0.188	0.000
70500.	0.400	0.355	0.277	0.124	0.000
77550.	0.355	0.307	0.224	0.061	0.000
84600.	0.312	0.261	0.173	0.001	0.000
91650.	0.269	0.216	0.123	0.000	0.000
98700.	0.228	0.172	0.074	0.000	0.000
105750.	0.188	0.129	0.027	0.000	0.000
112800.	0.149	0.088	0.000	0.000	0.000
119850.	0.110	0.047	0.000	0.000	0.000
126900.	0.073	0.007	0.000	0.000	0.000
133950.	0.036	0.000	0.000	0.000	0.000

etc...........

Figure 1. Calibration of Nickel Base Alloy 713C Elastic Modulus as a Function of Temperature

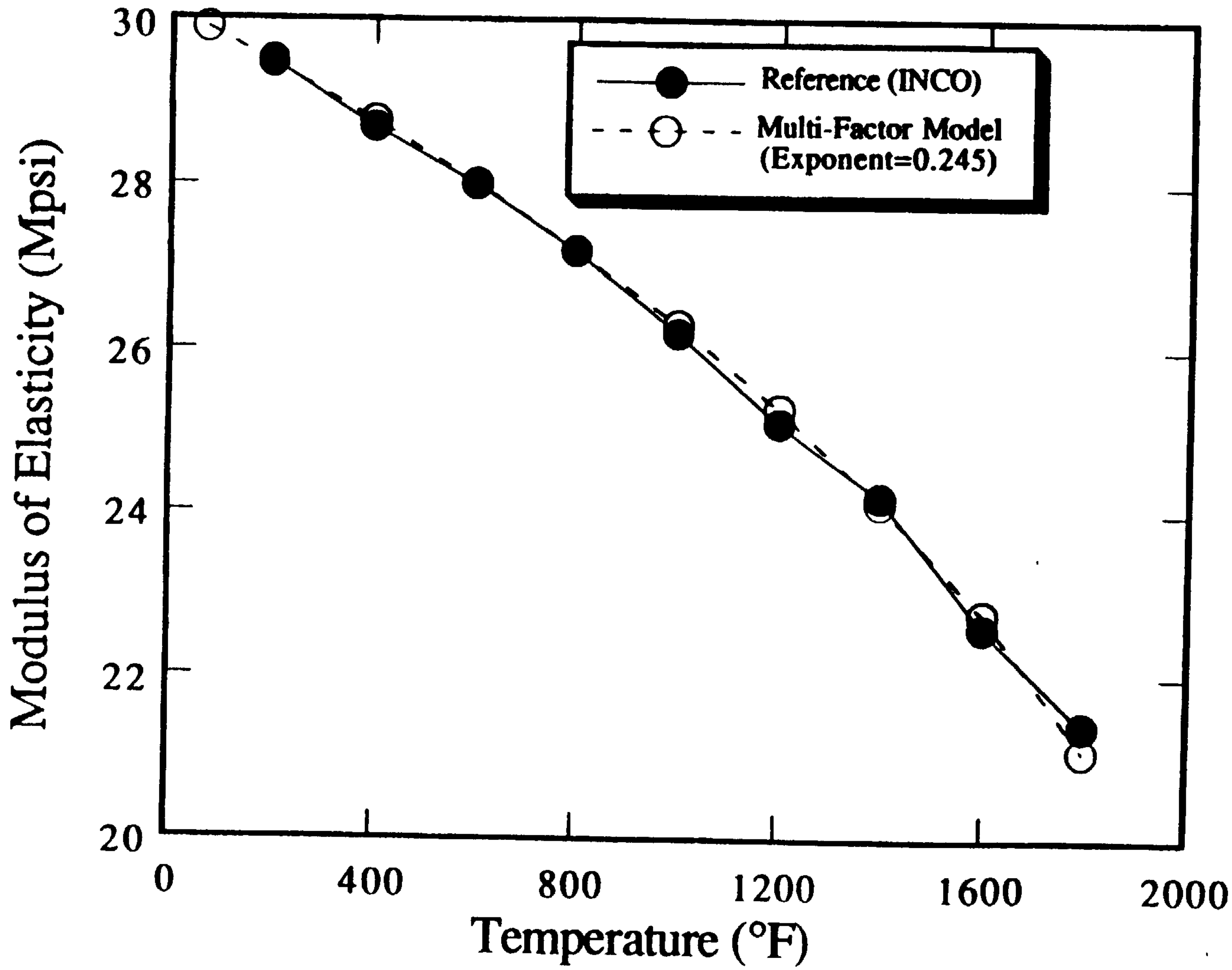


Figure 2. Calibration of Nickel Base Alloy IN-100 Coefficient of Thermal Expansion as a Function of Temperature

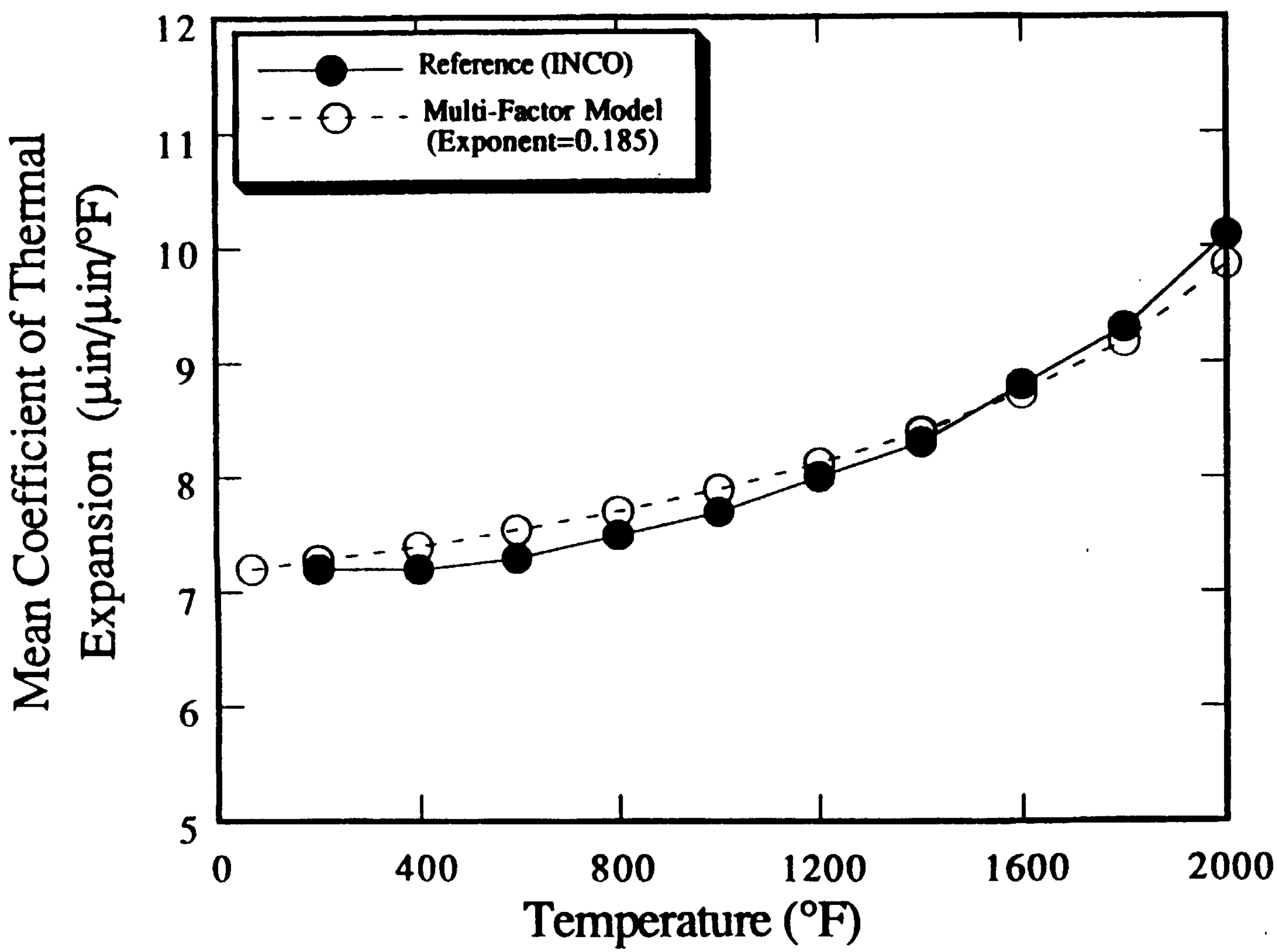


Figure 3. Calibration of Nickel Base Alloy MAR-M 200 Thermal Conductivity as a Function of Temperature

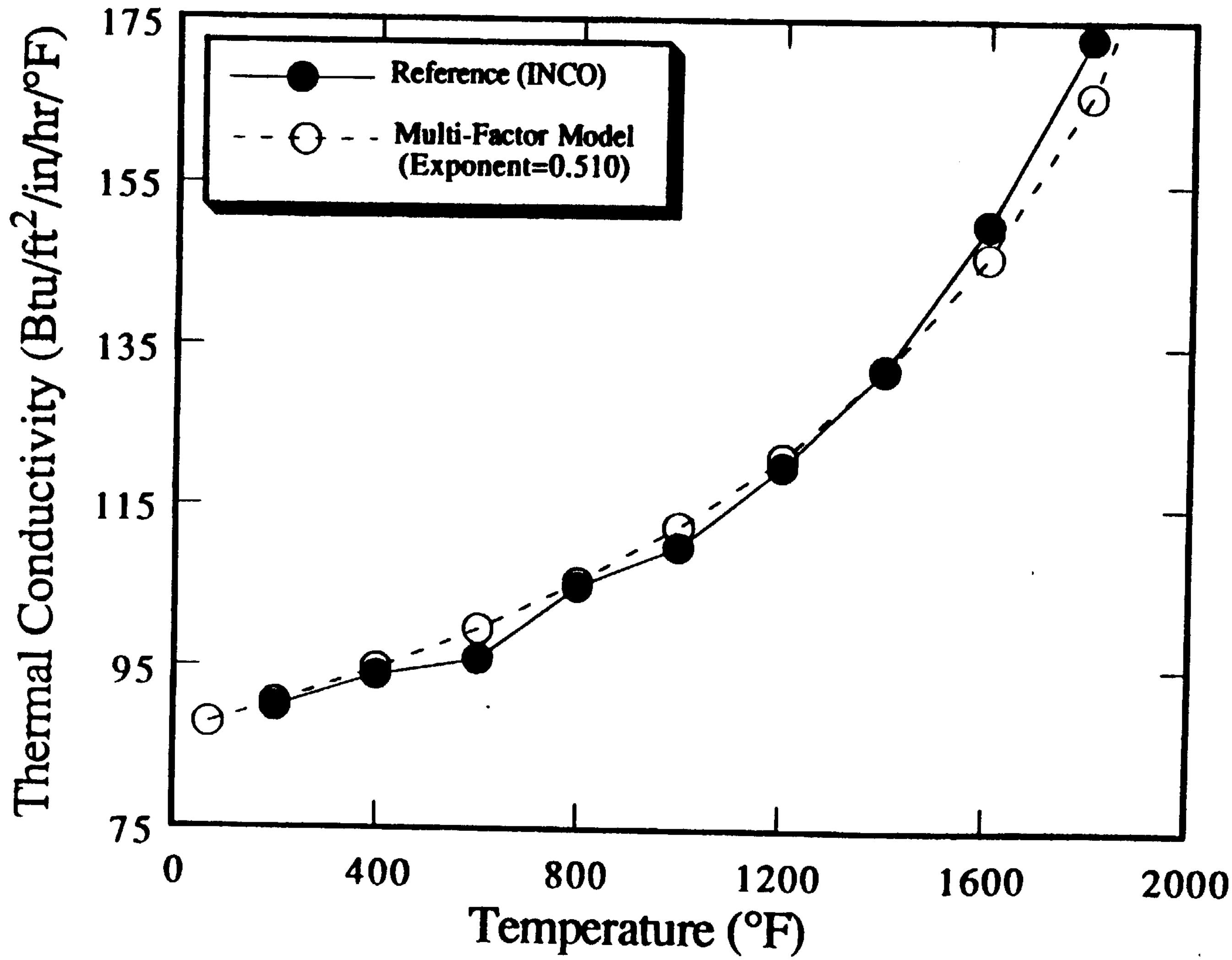


Figure 4. Calibration of Nickel Base Alloy IN-100 Specific Heat as a Function of Temperature

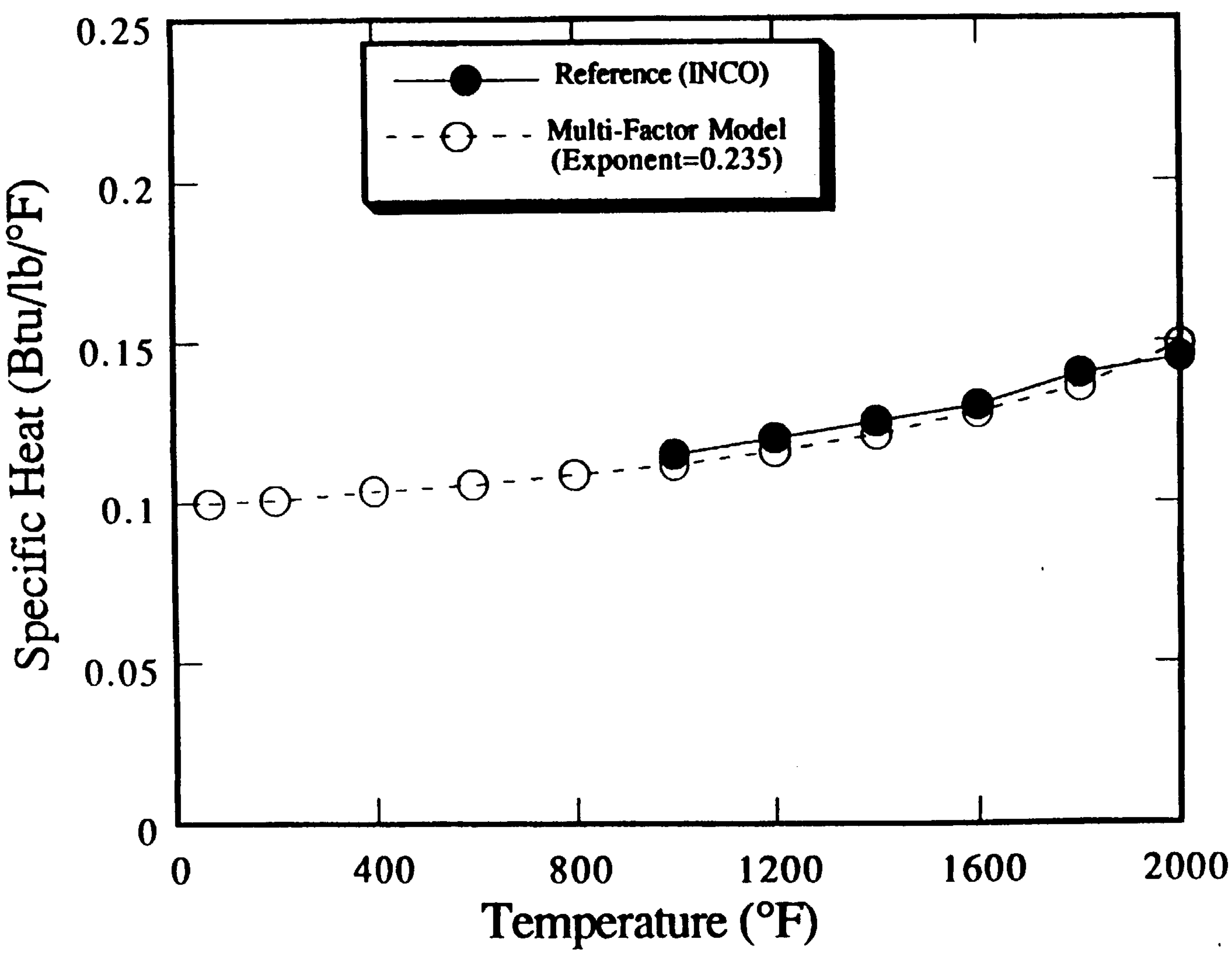


Figure 5. Calibration of Nickel Base Alloy B-1900 Tensile Elongation as a Function of Temperature

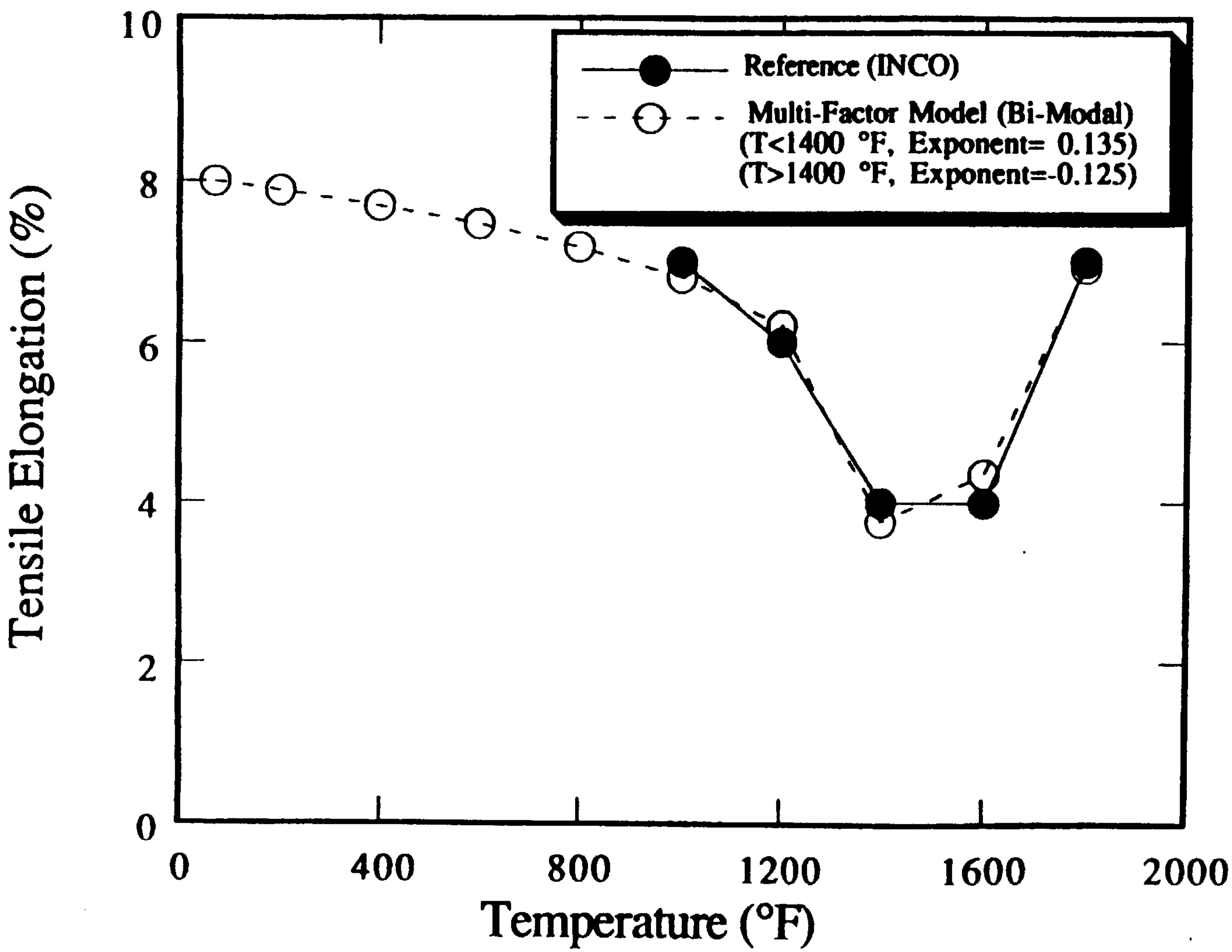


Figure 6. Calibration of Nickel Base Alloy B-1900 Yield Strength as a Function of Temperature

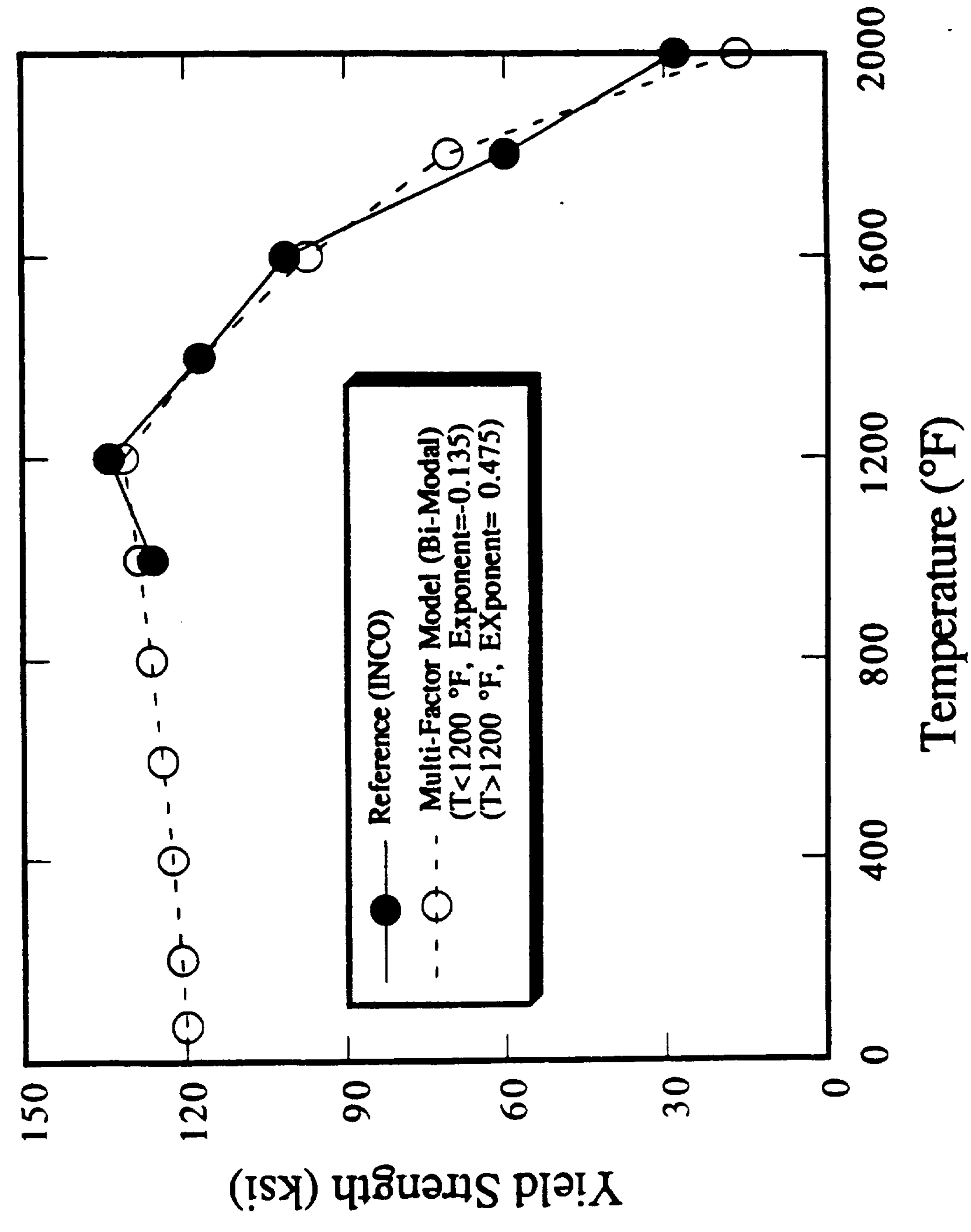
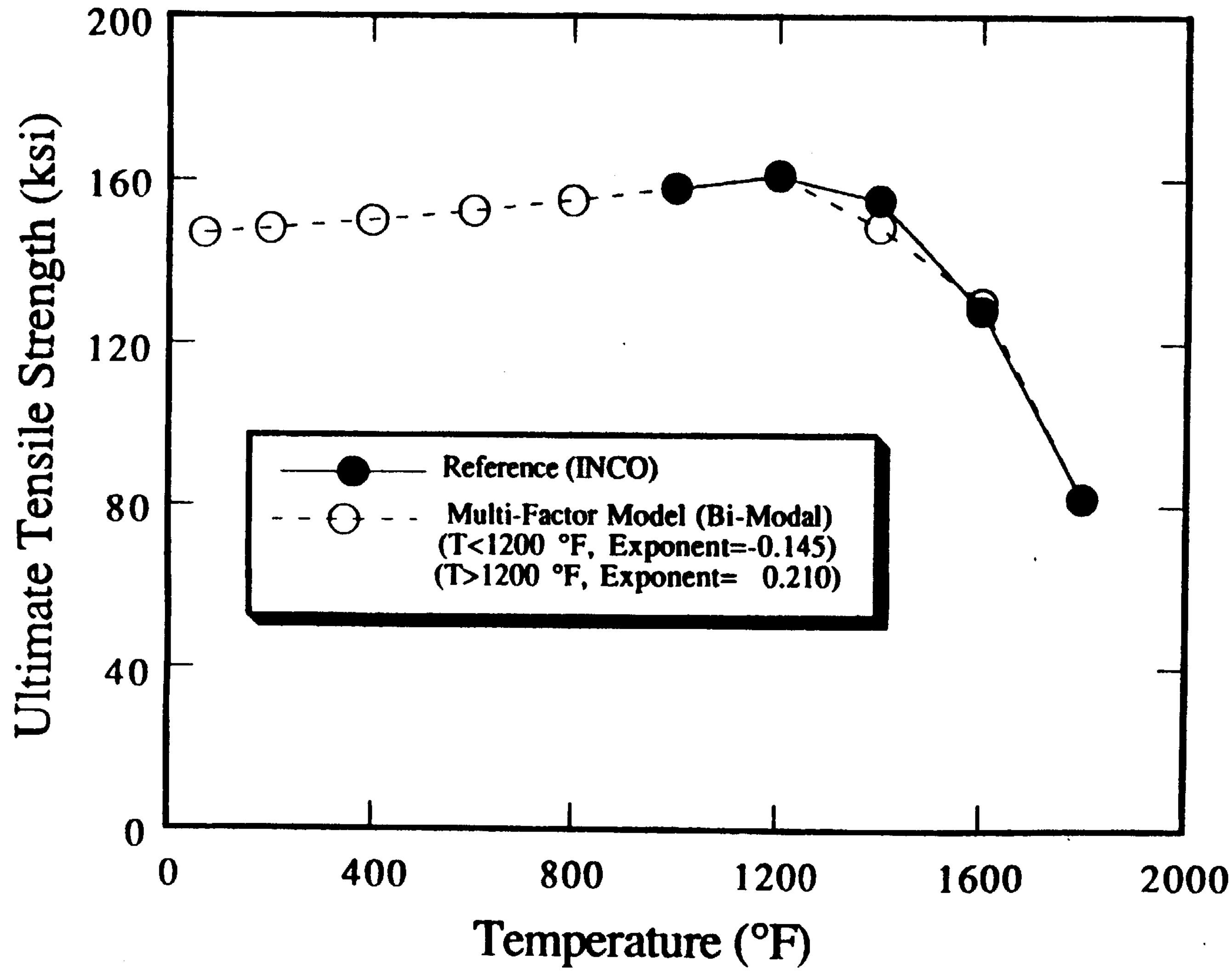


Figure 7. Calibration of Nickel Base Alloy IN-100 Ultimate Tensile Strength as a Function of Temperature



REPORT DOCUMENTATION PAGE	Form Approved OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	October 2001	Final Contractor Report

4. TITLE AND SUBTITLE

Engine Structural Analysis Software

5. FUNDING NUMBERS

WU–714–01–10–00
NAS3–26617
Task Order No. 40

6. AUTHOR(S)

R.L. McKnight, R.J. Maffeo, S. Schrantz, M.S. Hartle, G.S. Bechtel, K. Lewis, and M. Ridgway

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

GE Aircraft Engines
Advanced Mechanical Technology Operation
Cincinnati, Ohio 45215–6301

8. PERFORMING ORGANIZATION REPORT NUMBER

E–12822

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA CR—2001-210968

11. SUPPLEMENTARY NOTES

Contents were reproduced from the best available copy as provided by the authors. Project Manager, Christos C. Chamis, Research and Technology Directorate, NASA Glenn Research Center, organization code 5000, 216–433–3252.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Categories: 05 and 39 Distribution: Nonstandard

Available electronically at http://gltrs.grc.nasa.gov/GLTRS

This publication is available from the NASA Center for AeroSpace Information, 301–621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT (*Maximum 200 words*)

The report describes the technical effort to develop: (1) geometry recipes for nozzles, inlets, disks, frames, shafts, and ducts in finite element form, (2) component design tools for nozzles, inlets, disks, frames, shafts, and ducts which utilize the recipes and (3) an integrated design tool which combines the simulations of the nozzles, inlets, disks, frames, shafts, and ducts with the previously developed combustor, turbine blade, and turbine vane models for a total engine representation. These developments will be accomplished in cooperation and in conjunction with comparable efforts of NASA Glenn Research Center.

14. SUBJECT TERMS

Integrated analysis methods; Component geometry; Finite element analyses; Multiphase design simulations

15. NUMBER OF PAGES 208

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	

Figure 8. Calibration of Nickel Base Alloy B-1900
Rupture Strength at Various Temperatures
Sustained Stress = 90% of Static Strength