NASA-CR-198795

Ceramic Material Life Prediction - A Program to Translate ANSYS Results to CARES/LIFE Reliability Analysis

Final Report

Dr. Pieter Von Hermann/Adam Pintz

Period Covered: 3/1/88 to 4/26/94

Cleveland State University
1983 E. 24th Street
Cleveland, Ohio 44115

Grant Number: NAG 3-879

(NASA-CR-198795) CERAMIC MATERIAL LIFE PREDICTION: A PROGRAM TO TRANSLATE ANSYS RESULTS TO CARES/LIFE RELIABILITY ANALYSIS Final Report, 1 Mar. 1988 - 26 Apr. 1994 (Cleveland State Univ.) 29 p

N95-29367

Unclas

G3/27 0055114

ANSCARES

A Program Translating ANSYS Finite Element Results for CARES/LIFE Reliability Analysis

Lynn M. Powers* and John W. Spring†
Cleveland State University
Cleveland, OH 44115

Introduction

This manual describes the use of the ANSCARES program to prepare a neutral file of FEM stress results taken from ANSYS Release 5.0,[5] in the format needed by the CARES/LIFE[1] ceramics reliability program. It is intended for use by experienced users of ANSYS and CARES. Knowledge of compiling and linking FORTRAN programs is also required. ANSCARES has been jointly developed by NASA's Lewis Research Center and The Advanced Manufacturing Center at Cleveland State University.

Maximum use is made of existing routines (from other CARES interface programs and ANSYS routines) to extract the finite element results and prepare the neutral file for input to the reliability analysis. FORTRAN and machine language routines as described in Reference[6] are used to read the ANSYS results file. Sub-element stresses are computed and written to a neutral file using FORTRAN subroutines which are nearly identical to those used in the NASCARES (MSC/NASTRAN to CARES) interface.

* Resident Research Associate - NASA Lewis Research Center
† Advanced Manufacturing Center

Ceramics Reliability Analysis:

Increasing numbers of components are being made of ceramic materials in order to take advantage of higher temperature and corrosion resistance characteristics. Unfortunately, ceramic materials do not follow the same simple strength relations that characterize ductile metals. The CARES/LIFE (Ceramic Analysis and Reliability Evaluation of Structures Life Prediction Program)[1] program has been developed by NASA Lewis Research Center to predict the strength and reliability of ceramic components.

Probabilistic component design involves predicting the probability of failure for a thermomechanically loaded component from specimen rupture data. Typically these experiments are performed using many simple geometry flexural or tensile test specimens. A static, dynamic, or cyclic load is applied to each specimen until fracture. Statistical strength and SCG (fatigue) parameters are then determined from these data. Using these parameters and the results obtained from a finite element analysis, the time-dependent reliability for a complex component geometry and loading is then predicted. Appropriate design changes are made until an acceptable probability of failure has been reached. This design methodology combines the statistical nature of strength-controlling flaws with the mechanics of crack growth to allow for multiaxial stress states, concurrent (simultaneously occurring) flaw populations, and subcritical crack growth. These issues are addressed within the CARES/LIFE program.

The stress distribution within the component is provided by a file extracted from the results of a finite element method (FEM) calculation. Interfaces have been developed to the commercial FEM programs ABAQUS, Version 4.9-5.X (ABACARES)[2], MSC/NASTRAN version 67 (NASCARES)[3], and ANSYS, Release 4.4 (ANSYS/CARES)[4]. The ABAQUS and NASTRAN versions prepare a neutral file, in standardized format, which is accepted by the most recent reformulated CARES/LIFE program, while the ANSYS version uses the older concept of reading the FEM results through subroutines called from inside the CARES program. This current effort develops an interface that is compatible with the most recent versions of ANSYS and CARES/LIFE. A translator for ANSYS version 4.4A to CARES/LIFE is also available.

Neutral Data Base Concept

The concept of a formatted neutral data base allows for the interfacing of CARES/LIFE to several finite element packages. The results from the finite element analysis (available in the form of standard output or binary files and/or plot data files) have to be interpreted by an interface program. Due to the different formats used for output of the results in each finite element program, this interpreter program has to be adapted to the finite element software used.

The finite element results required for the reliability evaluation are assembled into the formatted neutral data base. A further advantage of this approach is the ease of transfer of this formatted neutral database (ASCII file) to different computer systems. Interpreter control parameters, such as the number of element groups to be considered in the reliability analysis, flags for accounting for temperature field gradients, etc., are usually input via an interpreter control file.

A subelement technique has been implemented. The reliability analysis is now performed at each Gauss integration point instead of using element averaging techniques. In the context of finite element analysis, stresses are determined at the Gaussian integration points where the local stiffness matrix has been evaluated. The subelement technique implies that each Gaussian integration point corresponds to a subelement. The subelement volume or area is defined as the contribution of the integration point to the element volume in the course of the numerical integration procedure.

The location of the Gaussian integration point in the natural space of the finite element, as well as corresponding weight functions have to be considered when the subelement volume is calculated. For this reason, the number of subelements in each element depends on the integration order chosen, and as a consequence, the element type. The increased number of points for stress evaluation accounts for the variation in stress over the element. Thus, considerable improvements in the accuracy of the reliability analysis have been realized.[3]

The neutral data base supplies the finite element results and data required for the reliability analysis, including the stress tensors and temperatures at the integration points. The structure of the neutral data base is optimized with respect to memory. The finite element data is arranged within the neutral data base using the following hierarchy: element groups, elements and subelements. The element group data contain information regarding the number of elements within the group. In addition to the element types available in the previous version of CARES (three-dimensional brick elements for volume flaw analysis, membrane elements for surface flaw analysis), two-dimensional finite elements are also implemented. By assuming plane stress conditions or exploiting symmetry of the structure, the size of the finite element model can be reduced using a two-dimensional mesh. Thus, the number of degrees of freedom could be substantially reduced.

The two-dimensional modeling options available (in plane loading vs. bending) depend on the method used to determine the volume. For ANSYS 5.0, the volume is calculated by multiplying the area of a subelement by the thickness of the element. Therefore, bending effects are not taken into account because the stress is assumed to be constant through the thickness.

Information pertaining to the elements include the number of subelements and the material identification number. Finally, the subelement group data contain information regarding subvolume, stress tensor (i.e., the stress tensor at the integration point), and subelement temperature. The stresses and temperatures are assumed to be constant over the subelement considered.

Based on the global hierarchy outlined above, the neutral file data is arranged into records with standard FORTRAN formats: the first record (format A80) specifies the title for the particular reliability evaluation. The next entry (format 2I5) contains the number of element groups (both volume as well as surface (shell) elements) to be considered in the reliability analysis and the a flag indicating combined volume as well as surface flaw reliability analysis for shell elements.

For every element groups, a loop is started with the next statement (format 4I5): the identification number of the element group considered, the element type corresponding to PATRAN element type codes, the number of elements within this element group, and a shell element type indicator have to be input. If shell elements are used in the analysis, the indicator has to be set to 1. Information pertaining to the element level is read in a loop over the number of elements within the element group considered (format 4I5,3F15.0).

The element level data contain the identification number of the element considered, the number of subelements within this element, the material identification number for volume flaw analysis, the material identification number for surface flaw analysis, the element volume, the element thickness only in case of shell elements, and the averaged temperature of this element. The identification number of the element should be the same as the corresponding element number in the finite element mesh, if further postprocessing of the reliability results are planned. The number of subelements within this element depends on the Gaussian integration order chosen in the finite element analysis and on the element type. If stress data is only available at the center of the element (standard case in NASTRAN analyses prepared for previous CARES versions), one subelement is specified. The material identification numbers must correspond to the material numbers used in the finite element analysis as well as the material numbers on the control file (containing the Weibull parameters). Element volume and element temperature are necessary only for element group information summaries in the printout file. For the reliability evaluation the subelement data are directly used.

A third loop is started for reading the subelement level data. For volume type elements the identification number of the subelement, the subelement volume, the subelement temperature and the full stress tensor at the integration point (defined by σ_{xx}, σ_{yy}, σ_{zz}, σ_{xy}, σ_{yz} and σ_{xz} in the local coordinate system) have to be specified (format I5,5F15.0,/,35X,3F15.0).

For shell type elements plane stress conditions are assumed. For that reason only σ_{xx}, σ_{yy} and σ_{xy} are necessary for the definition of the subelement stress tensor. Instead of the subelement volume, the subelement area has to be specified (I5,5F15.0).

Typically, subelement volumes are not included with standard finite element output. Thus, the volume of each subelement (corresponding to a Gauss integration point) is calculated in the interpreter program using the shape functions inherent to the element type. In the usual context of finite element methods, the volume of a three-dimensional element, (i. e., brick, wedge, pyramid or tetrahedron) is calculated after transformation into the natural coordinate space,

$$V = \int_{-1}^{1} \int_{-1}^{1} \int_{-1}^{1} \det \mathbf{J}(r,s,t)\; dr\; ds\; dt \tag{1}$$

where **J** is the Jacobian operator and r,s,t are the natural coordinates. By applying a Gaussian integration scheme, the integral can be expressed as

$$V = \sum_{i=1}^{n} \sum_{j=1}^{n} \sum_{k=1}^{n} \det \mathbf{J}(r_i,s_j,t_k)\ W_i\ W_j\ W_k \tag{2}$$

where n is the integration order, r_i,s_j,t_k are the coordinates of the Gaussian integration points in natural space and the W's are the weight functions. The area of an element (i. e., quadrilateral or triangle) is determined in a similar manner

$$A = \sum_{i=1}^{n} \sum_{j=1}^{n} \det \mathbf{J}(r_i,s_j)\ W_i\ W_j \tag{3}$$

The volume of an axisymmetric element can also be determined similarly

$$V = R \sum_{i=1}^{n} \sum_{j=1}^{n} \det \mathbf{J}(r_i,s_j)\ W_i\ W_j \tag{4}$$

where R is the radial coordinate of the integration point in the global coordinate system.

ANSYS/CARES Analysis Procedure:

Fracture probability prediction on a ceramic part begins with the execution of the ANSYS program using a finite element model of the part in question. This model will typically consist of volume elements with shell elements attached at the surfaces to provide both volume and surface data to CARES/LIFE. Supported element types are listed in Table 1. Shell elements should have membrane properties only and thicknesses near zero to avoid affecting the stiffness of the component. It is vital that the material identifications used in the ANSYS element descriptions agree exactly with those contained in the CARES/LIFE template file (MATID keyword in the Material Control Input).

Standard results utilized from the ANSYS linear elastic analysis results file include nodal coordinates, temperatures, and stress components at all nodes. The ANSCARES program is then used to compute subelement stresses, volumes, and temperatures as needed by the CARES/LIFE program. All needed data are read from the ANSYS results file (FILE.RST), and no user input is required except to change file naming and analysis type defaults, if desired. ANSCARES interprets one load case per run, therefore ANSCARES does not handle multiple load cases.

The final step is to execute the CARES/LIFE program using the neutral file prepared by ANSCARES and a template file containing material fracture properties. The template file also controls the type of analysis and the results computed. Figure 1 shows a flow diagram of a typical reliability analysis using finite element stress and volume results.

Installing ANSCARES:

The ANSCARES program is provided on disk with an example problem. ANSCARES consists of a main program and nine subroutines, all supplied in FORTRAN 77 source code in two modules called ANSCARES1.FOR and ANSCARES2.FOR (anscares1.f and anscares2.f: renamed for unix systems). This module should be copied into a suitable directory and compiled.

ANSCARES makes use of system-dependent, machine language subroutines included with the ANSYS distribution tape. These routines are designed to read data from the ANSYS results file. The specific subroutines needed are BINCLO, BININI, BINIQR, BINRD, BINSET, EXINC4, and SYSIQR, all of which are included in the machine language file BINLIB.OLB. This file must be copied from the ANSYS distribution tape to the directory used to run the ANSCARES program.

These machine language ANSYS routines must be linked with the compiled FORTRAN object code to create an executable version of ANSCARES. If they are all in the same library, the command to do this on a VAX/VMS computer is

```
LINK ANSCARES,BINLIB/LIBRARY
```

for a silicon graphics and possibly other unix systems, the command is

```
f77 -mp -g -o anscares anscares1.o anscares2.o /ansys50/objects/binlib.a
```

The options for the f77 command are those which were used to link the program on a SGI Personal IRIS. Linking operations are computer specific, and can be expected to make use of a different command on other computer systems. However, these are described in the appropriate ANSYS INSTALLATION GUIDE for your system under Stand-Alone Utility Programs, BINLIB Subroutines.

The ANSCARES disk also contains the ANSYS input file for the example problem, below, along with the corresponding CARES/LIFE template file. To confirm the proper installation, ANSYS, ANSCARES, and CARES/LIFE should be executed in turn using these files. Copies of the resulting ANSCARES output neutral file (ANSCARES.NEU) and CARES/LIFE output (CARES.OUT) are included for comparison.

ANSYS considerations:

At present, analysis employing eight ANSYS element types has been implemented into the ANSCARES program. These element types allow three-dimensional and axisymmetric analyses. Table 1 describes the ANSYS Release 5.0 element types supported by the ANSCARES program. To provide the stress results needed by ANSCARES, all element types must have the element description variable KEYOPT(5) set to a value of two (2). Element types that use the variable KEYOPT(6) should have its value remain at the default value of zero (0).

ANSCARES program execution

The ANSCARES program will interactively prompt for the names of the ANSYS stress and geometry results file to be read and the CARES neutral file to be generated. Depressing the ENTER key for each prompt will accept the default names of "FILE.RST" and "CARES.NEU". Additional prompts are for element or subelement analysis and warning/progress messages to be sent to the monitor or to a file. Subelement analysis is highly recommended. Again pressing the ENTER key will accept the default values of subelement analysis (ICARES=2) and display of messages on the monitor. A response of "n" to the prompt will allow the user to specify that messages are saved in the "ANSCARES.OUT" file instead, when IOUT is set to 9 instead of (default) 6. Depending on the computer system, it may be necessary to change the default value for monitor display to some value different from 6.

There is no specific limit on the number of nodes and elements permitted in the finite element model. All needed node and element data is stored in a general purpose array (the A array). The maximum number of nodes and elements is limited only by the amount of space available in this array. Of course, some element types require more data for each element and use more space than others. The declared size of the array can be increased or decreased as necessary. The maximum size of the A array is set by the parameter MTOT in the ANSCARES program (main routine) and is related to the amount of computer resources available. After changing the MTOT parameter, the ANSCARES main program, but not necessarily the subroutines, must be re-compiled and all programs re-linked to establish the new limits.

The parameter ITOT in the ANSCARES program (main program) is used to define the size of the temporary buffer used when reading the ANSYS results file. The ANSYS results variable RECLNG contains the minimum size of buffer needed. ITOT is currently set to the size needed by the VAX computer version of ANSYS (7680 words). The ANSCARES program will print a warning message and exit if the computer system in use requires more space. To minimize computer storage it is necessary to determine the minimum buffer needed. This can be accomplished by setting the ITOT variable to 1 which will result in a message giving the required size when ANSCARES is executed. ITOT can then be given that value. As with MTOT, after changing the ITOT parameter the ANSCARES main program must be re-compiled and all programs re-linked to establish the new buffer size.

ANSCARES Program Description:

Machine language routines supplied with the ANSYS program are called as needed to read the ANSYS analysis results file. This file is described in Chapter 6 of Ref [6]. The routines used to access this file are documented in Chapter 5, also in Ref [6].

The ANSCARES main program serves to establish sizes for arrays, call the subroutines that read the ANSYS results file, and call the subroutines that prepare the CARES/LIFE neutral file. Figure 2 shows the major functions of the various modules in the ANSCARES program. ANSYS results are read by the subroutines EANSYS, RDGEOM, and RDSOLU. These three routines are based on and remain very similar to the example programs beginning on page A-21 of Ref. [6].

The EANSYS subroutine opens the ANSYS results file (FILE.RST) and determines the number of nodes and elements in the model, and other parameters. Subroutine RDGEOM is then called by EANSYS to extract coordinate locations, element connectivity, and properties. This data is stored in the A array for later use. A count is also made of the various kinds of elements in the model. EANSYS then calls subroutine RDSOLU to extract stress and temperature results, which are also stored in the A array. This completes reading of the data from the ANSYS results file.

The HEXA, PENTA, TETRA, QUAD8, TRIA6, and TRIAX subroutines are then called by ANSCARES as needed, depending on the kinds of elements in the model. These subroutines are very similar to the same-named subroutines in the previous NASCARES program.[3]

Example Problem:

Sample problems were executed using the ANSYS-ANSCARES-CARES/LIFE sequence to verify the accuracy of the analysis as compared to interfaces for earlier versions of ANSYS. Models including elements with and without mid-side nodes and using solid, shell, and axisymmetric element types confirmed program operation. One of the models, EXAMPLE1 from Reference 4, is described below.

EXAMPLE1 is a transversely loaded ceramic circular plate. In ANSYS this is treated as a sector suitably restrained to preserve symmetry. The plate is subdivided into 48 equal sectors, making each sector 7.5° wide. The model of the sector is divided into 40 three-dimensional solid elements and 10 shell elements, all with mid-side nodes. The four solid elements and the shell element at the center of the model use duplicated node numbers to make five sided prism solid and triangular shell elements. The remaining 36 solid and 9 shell elements remain as solids with six sides and shells with four edges.

The circular plate has an outside diameter of 2.00 inches and a thickness of 0.071 inches. Figure 3 is a side view of the sector, showing the element numbers. The four layers shown are made up of the solid elements, with the ten shell elements attached to the bottom of the lowest layer. Figures 4 and 5 show additional details of the node and element numbering. The sector is simply supported between elements 9 and 10 (equivalent to a ring support at a radius of 0.92 inches). A uniform pressure of 220 psi is applied to elements 31 through 39. Temperature is a uniform 70°F in all elements. Material properties used in this problem are given below:

- Material Number = 300 (Must agree with CARES template)
- Young's Modulus = 5.87×10^7 psi
- Density = 0.5 pounds/cubic inch
- Poisson's Ratio = 0.25

The entire ANSYS input file is included in Appendix A.

This example is provided to demonstrate model generation and neutral file design. The CARES/LIFE template file is supplied to run CARES/LIFE using the generated neutral file. The background for this example is described in reference [7]: Example 1 - Transversely Loaded Circular Disk.

ANSYS Element Type	Number of Nodes	Description	Comment
42-PLANE	4	Axisymmetric - 4 Edges	Not supported
	3	Axisymmetric - 3 Edges	KEYOPT(3) = 1
45-SOLID	8	Brick - 6 Sides	
	6	Prism - 5 Sides	
	4	Tetrahedron - 4 Sides	
63-SHELL	4	Shell - 4 Edges	
	3	Shell - 3 Edges	KEYOPT(1) = 0
72-SOLID	4	Tetrahedron - 4 Sides	
82-PLANE	8	Axisymmetric - 4 Edges	Not Supported
	6	Axisymmetric - 3 Edges	KEYOPT(3) = 1
92-SOLID	10	Tetrahedron - 4 Sides	
93-SHELL	4	Shell - 4 Edges	
	3	Shell - 3 Edges	
95-SOLID	20	Brick - 6 Sides	
	15	Prism - 5 Sides	
	13	Pyramid - 5 Sides	Not Supported
	10	Tetrahedron - 4 Sides	

Note: For all of the ANSYS element types above:
KEYOPT(5) + 2
KEYOPT(6) = 0

Table 1 - ANSYS element types supported by ANSCARES

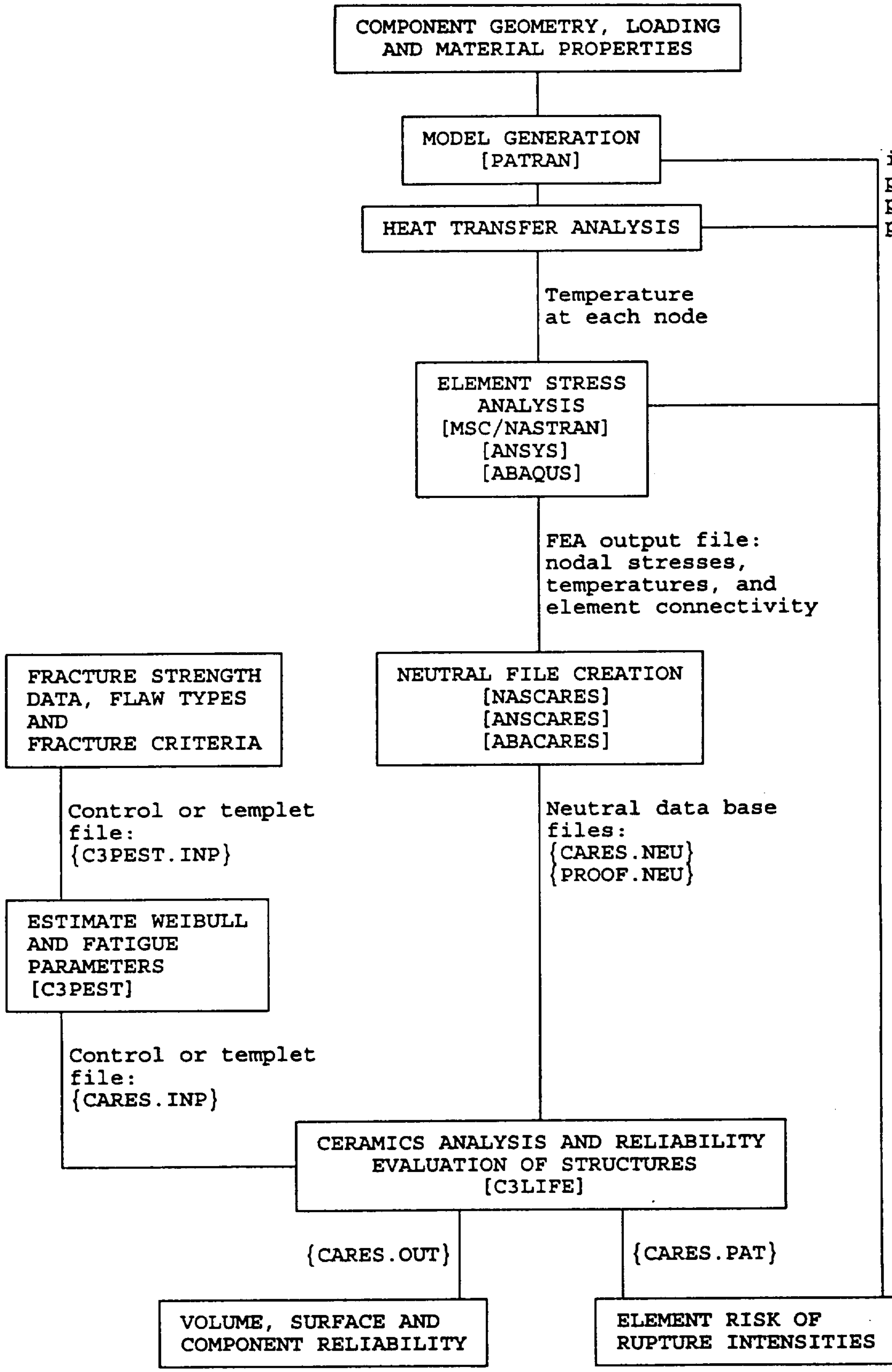

Figure 1 - Block diagram for the analysis and time-dependent reliability evaluation of ceramic components. This analysis includes the CARES/LIFE integrated design program, finite element analysis and, PATRAN PLUS pre- and post-processing.

```
Begin ANSCARES program

Check for ANSYS results file

Call EANSYS Subroutine
        Extract disk pointers and model parameters

        Call RDGEOM Subroutine
                Extract and save node coordinates
                For each element:
                        Extract and save element properties
                        Extract element connectivity
                                Save in temporary space
                        Add to element type counter
                For each element type:
                        Eliminate duplicate element nodes
                        Move connectivity to save array

        Call RDSOLU Subroutine
                For each element:
                        Extract and save element stresses
                For each element:
                        Extract temperatures at nodes
                        Save temperatures in node order

Write neutral file header

For each element type in the model:

        Call appropriate element type subroutine
                        (HEXA, PENTA, TETRA, QUAD8, TRIA6, or TRIAX)
                For each element of this type:
                        Compute sub-element stresses
                        Compute sub-element volumes
                        Compute average temperature
                        Write results to neutral file

Close files

End ANSCARES program
```

Figure 2 - ANSCARES Program Logic

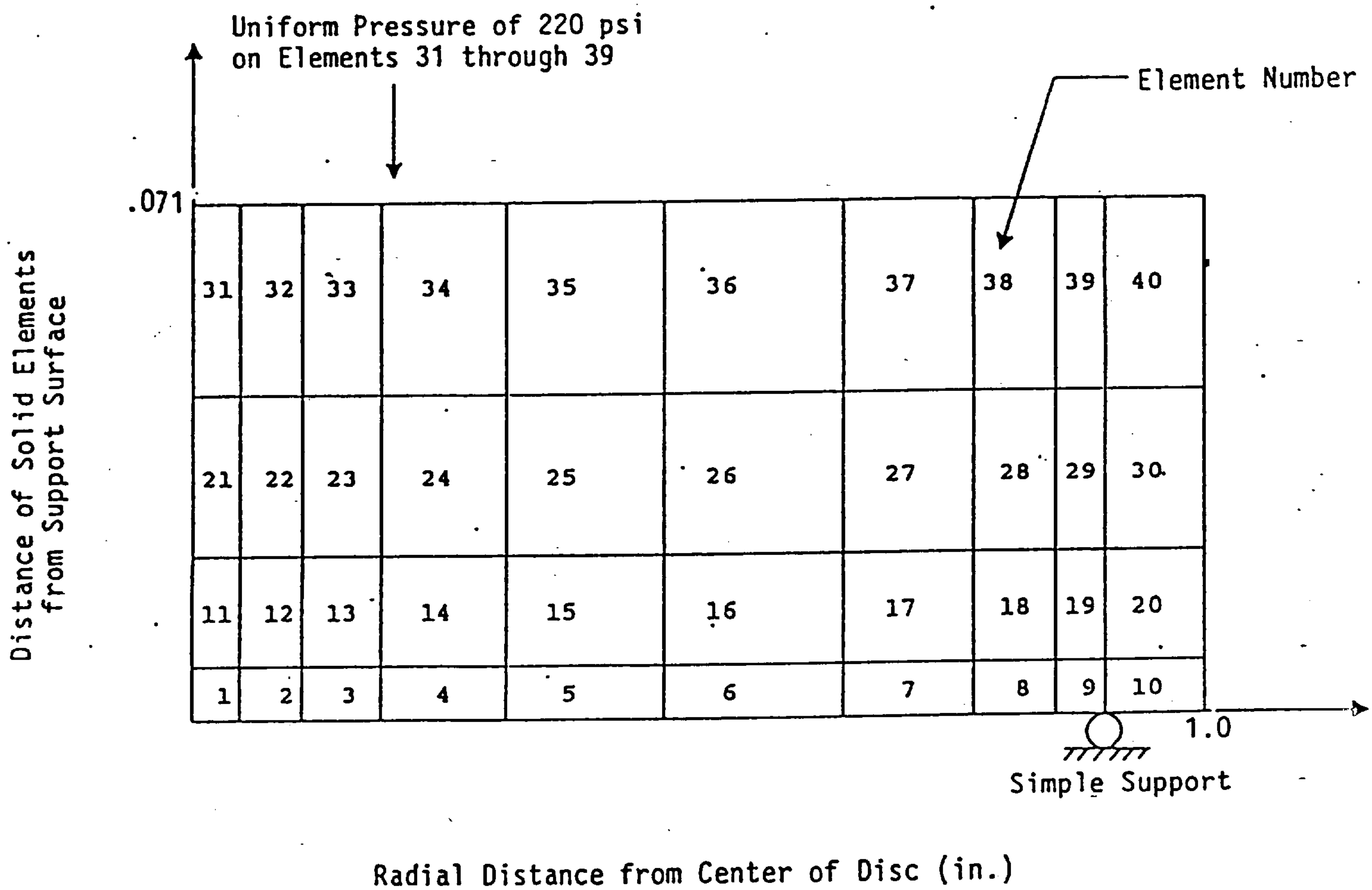

Figure 3 - Side view of a sector of the ceramic plate

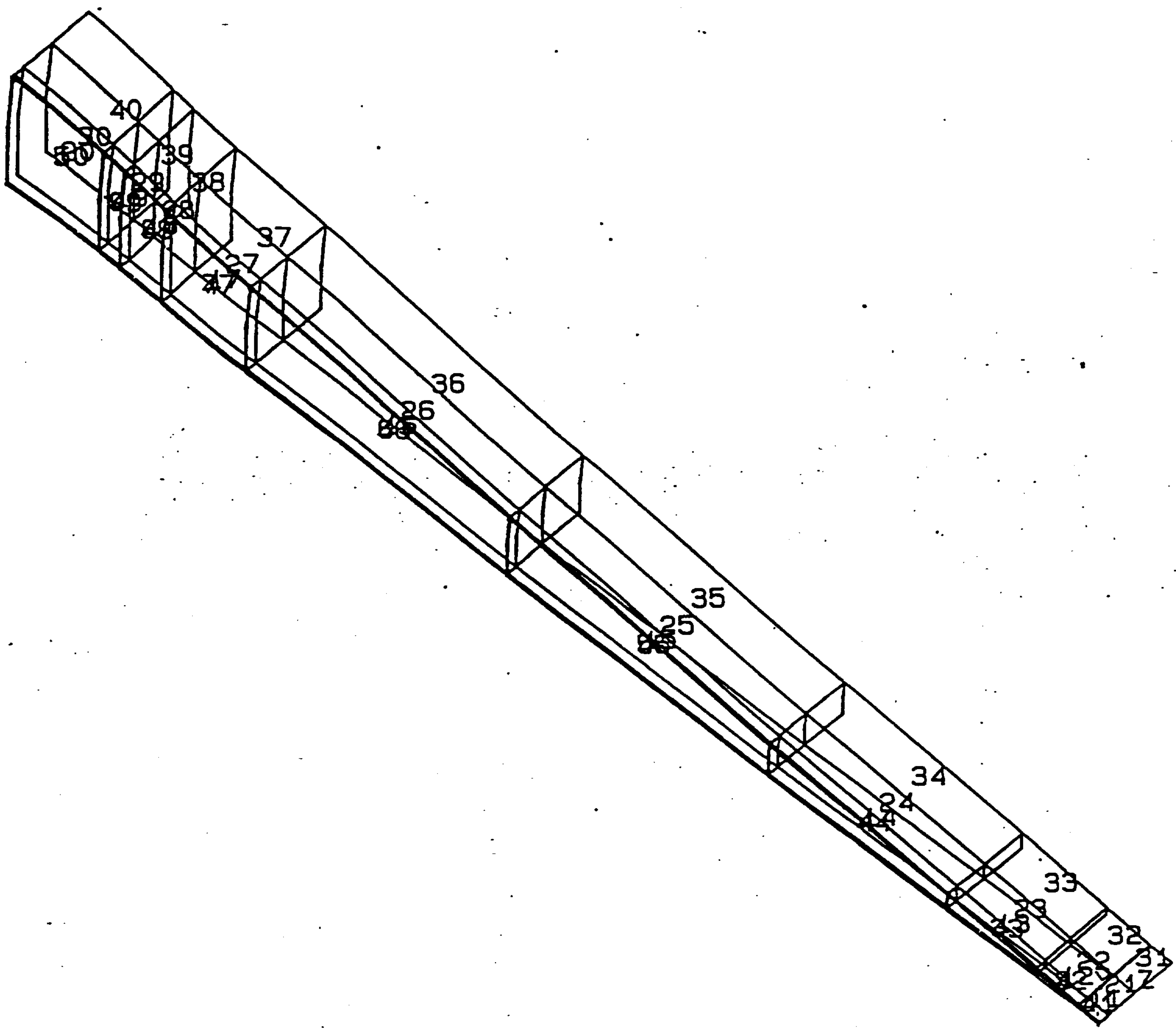

Figure 4 - Solid elements in the example

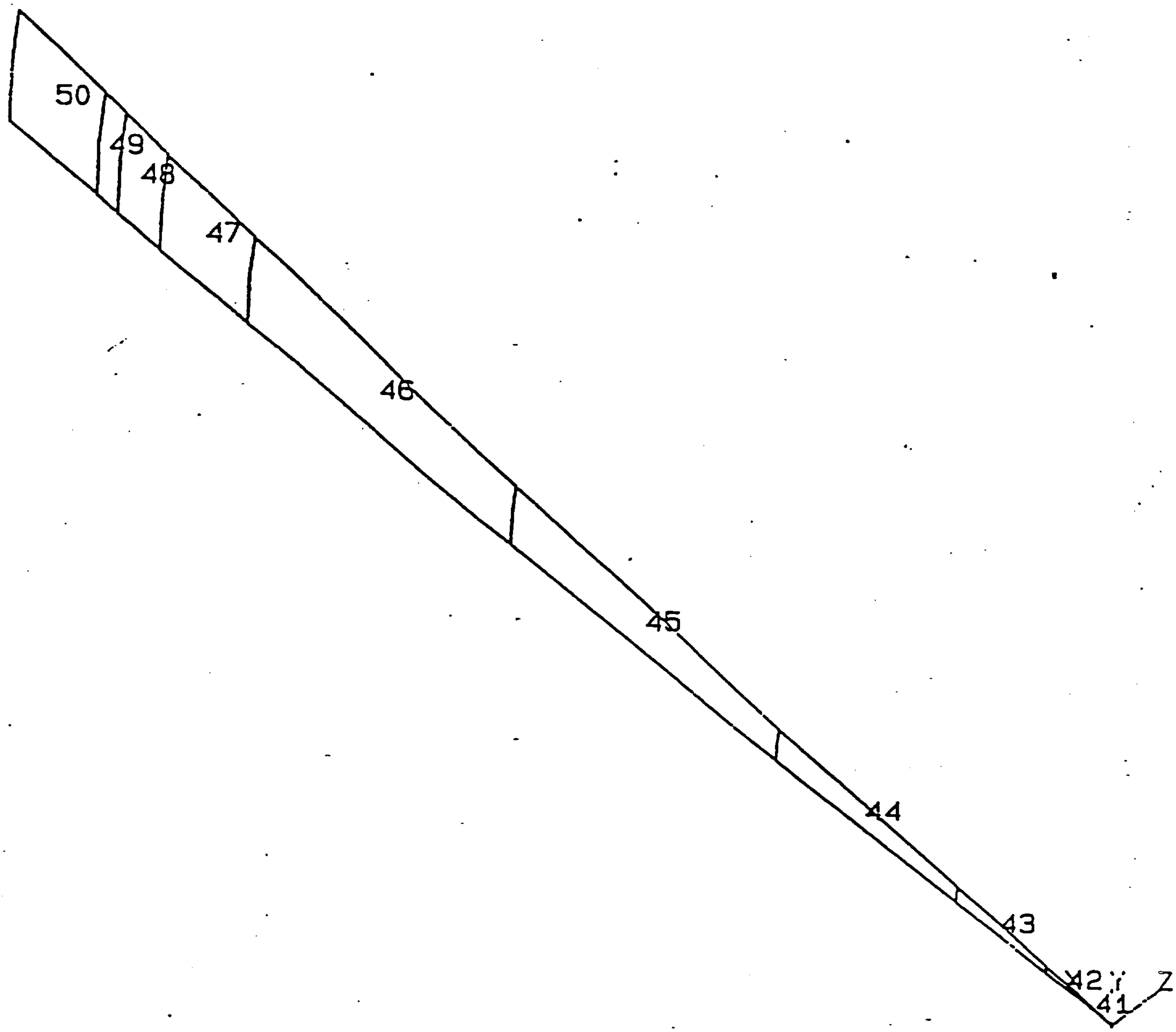

Figure 5 - Element plot of bottom shell layer

References

[1] Nemeth, N.N., Powers, L.M., Janosik, L.A., and Gyekenyesi, J.P., "Ceramic Analysis and Reliability Evaluation of Structures Life Prediction Program (CARES/LIFE) Users and Programmers Manual", NASA TM-106316, to be published.

[2] Edwards, M.J., 'ABACARES User's Manual", Babcock & Wilcox, Research and Development Division, Alliance, Ohio, 44601, January 7, 1993.

[3] Powers, L.M., Starlinger, A., and Gyekenyesi, J.P., "Ceramic Component Reliability With the Restructured NASA/Cares Computer Program", Presented at the 37th International Gas Turbine and Aeroengine Congress, Sponsored by ASME, Cologne, Germany, June 1-4, 1992, ASME Paper No. 92-GT-383.

[4] Pintz, A., Abumeri, G.H., and Manderscheid, J.M., "User's Manual - NASA/CARES program Using ANSYS as a Pre-Processor Program", July, 1989.

[5] Swanson Analysis Systems Inc., ANSYS Rev. 5.0 Engineering Analysis System User's Manual Vol I, II, and III.

[6] Swanson Analysis Systems Inc., ANSYS Programmer's Manual for Rev. 5.0, Beta Version Draft, May 11, 1992.

[7] Nemeth, N.N., Manderscheid, J.M., and Gyekenyesi, J.P., "Ceramic Analysis and Reliability Evaluation of Structures (CARES)", NASA TP-2916, Aug. 1990.

Appendix A

ANSYS Input File for Example1

```
/BATCH,LIST
/PREP7
/TITLE, STRESS ANALYSIS OF A SIMPLY SUPPORTED DISC, UNIQUE ELEMENT PRESSURE
ET,1,95
KEYOPT,1,5,2
ET,2,93
KEYOPT,2,5,2
EX,300,5.87E07
DENS,300,0.5
NUXY,300,0.25
TREF,70
csys,1
N,1,0.,0.,0.
N,2,0.01,0.,0.
N,3,0.02,0.,0.
N,4,0.04,0.,0.
N,5,0.06,0.,0.
N,6,0.10,0.,0.
N,7,0.14,0.,0.
N,8,0.22,0.,0.
N,9,0.30,0.,0.
N,10,0.42,0.,0.
N,11,0.54,0.,0.
N,12,0.66,0.,0.
N,13,0.78,0.,0.
N,14,0.82,0.,0.
N,15,0.86,0.,0.
N,16,0.88,0.,0.
N,17,0.90,0.,0.
N,18,0.91,0.,0.
N,19,0.92,0.,0.
N,20,0.96,0.,0.
N,21,1.00,0.,0.
NGEN,2,72,1,21,1,,,0.002
NGEN,2,72,73,93,1,,,0.008
NGEN,2,72,145,165,1,,,0.0255
NGEN,2,72,217,237,1,,,0.0355
N,52,0.,0.,0.001
N,53,0.02,0.,0.001
N,54,0.06,0.,0.001
N,55,0.14,0.,0.001
N,56,0.30,0.,0.001
N,57,0.54,0.,0.001
N,58,0.78,0.,0.001
N,59,0.86,0.,0.001
N,60,0.90,0.,0.001
N,61,0.92,0.,0.001
N,62,1.00,0.,0.001
NGEN,2,72,52,62,1,,,0.005
NGEN,2,72,124,134,1,,,0.01675
```

```
NGEN,2,72,196,206,1,,,0.03025
N,22,0.02,3.75,0.
N,23,0.06,3.75,0.
N,24,0.14,3.75,0.
N,25,0.30,3.75,0.
N,26,0.54,3.75,0.
N,27,0.78,3.75,0.
N,28,0.86,3.75,0.
N,29,0.90,3.75,0.
N,30,0.92,3.75,0.
N,31,1.00,3.75,0.
NGEN,2,72,22,31,1,,,0.002
NGEN,2,72,94,103,1,,,0.008
NGEN,2,72,166,175,1,,,0.0255
NGEN,2,72,238,247,1,,,0.0355
NGEN,2,30,2,21,1,,7.5,
NGEN,2,72,32,51,1,,,0.002
NGEN,2,72,104,123,1,,,0.008
NGEN,2,72,176,195,1,,,0.0255
NGEN,2,72,248,267,1,,,0.0355
NGEN,2,10,53,62,1,,7.5,
NGEN,2,72,63,72,1,,,0.005
NGEN,2,72,135,144,1,,,0.01675
NGEN,2,72,207,216,1,,,0.03025
nrot,1,339,1
TYPE,1
MAT,300
E,1,3,33,33,73,75,105,105
EMORE,2,22,33,32,74,94,105,104
EMORE,52,53,63,63
E,3,5,35,33,75,77,107,105
EMORE,4,23,34,22,76,95,106,94
EMORE,53,54,64,63
E,5,7,37,35,77,79,109,107
EMORE,6,24,36,23,78,96,108,95
EMORE,54,55,65,64
E,7,9,39,37,79,81,111,109
EMORE,8,25,38,24,80,97,110,96
EMORE,55,56,66,65
E,9,11,41,39,81,83,113,111
EMORE,10,26,40,25,82,98,112,97
EMORE,56,57,67,66
E,11,13,43,41,83,85,115,113
EMORE,12,27,42,26,84,99,114,98
EMORE,57,58,68,67
E,13,15,45,43,85,87,117,115
EMORE,14,28,44,27,86,100,116,99
EMORE,58,59,69,68
E,15,17,47,45,87,89,119,117
EMORE,16,29,46,28,88,101,118,100
EMORE,59,60,70,69
E,17,19,49,47,89,91,121,119
EMORE,18,30,48,29,90,102,120,101
EMORE,60,61,71,70
E,19,21,51,49,91,93,123,121
EMORE,20,31,50,30,92,103,122,102
```

EMORE,61,62,72,71
E,73,75,105,105,145,147,177,177
EMORE,74,94,105,104,146,166,177,176
EMORE,124,125,135,135
E,75,77,107,105,147,149,179,177
EMORE,76,95,106,94,148,167,178,166
EMORE,125,126,136,135
E,77,79,109,107,149,151,181,179
EMORE,78,96,108,95,150,168,180,167
EMORE,126,127,137,136
E,79,81,111,109,151,153,183,181
EMORE,80,97,110,96,152,169,182,168
EMORE,127,128,138,137
E,81,83,113,111,153,155,185,183
EMORE,82,98,112,97,154,170,184,169
EMORE,128,129,139,138
E,83,85,115,113,155,157,187,185
EMORE,84,99,114,98,156,171,186,170
EMORE,129,130,140,139
E,85,87,117,115,157,159,189,187
EMORE,86,100,116,99,158,172,188,171
EMORE,130,131,141,140
E,87,89,119,117,159,161,191,189
EMORE,88,101,118,100,160,173,190,172
EMORE,131,132,142,141
E,89,91,121,119,161,163,193,191
EMORE,90,102,120,101,162,174,192,173
EMORE,132,133,143,142
E,91,93,123,121,163,165,195,193
EMORE,92,103,122,102,164,175,194,174
EMORE,133,134,144,143
E,145,147,177,177,217,219,249,249
EMORE,146,166,177,176,218,238,249,248
EMORE,196,197,207,207
E,147,149,179,177,219,221,251,249
EMORE,148,167,178,166,220,239,250,238
EMORE,197,198,208,207
E,149,151,181,179,221,223,253,251
EMORE,150,168,180,167,222,240,252,239
EMORE,198,199,209,208
E,151,153,183,181,223,225,255,253
EMORE,152,169,182,168,224,241,254,240
EMORE,199,200,210,209
E,153,155,185,183,225,227,257,255
EMORE,154,170,184,169,226,242,256,241
EMORE,200,201,211,210
E,155,157,187,185,227,229,259,257
EMORE,156,171,186,170,228,243,258,242
EMORE,201,202,212,211
E,157,159,189,187,229,231,261,259
EMORE,158,172,188,171,230,244,260,243
EMORE,202,203,213,212
E,159,161,191,189,231,233,263,261
EMORE,160,173,190,172,232,245,262,244
EMORE,203,204,214,213
E,161,163,193,191,233,235,265,263

```
EMORE,162,174,192,173,234,246,264,245
EMORE,204,205,215,214
E,163,165,195,193,235,237,267,265
EMORE,164,175,194,174,236,247,266,246
EMORE,205,206,216,215
E,217,219,249,249,289,291,321,321
EMORE,218,238,249,248,290,310,321,320
EMORE,268,269,279,279
E,219,221,251,249,291,293,323,321
EMORE,220,239,250,238,292,311,322,310
EMORE,269,270,280,279
E,221,223,253,251,293,295,325,323
EMORE,222,240,252,239,294,312,324,311
EMORE,270,271,281,280
E,223,225,255,253,295,297,327,325
EMORE,224,241,254,240,296,313,326,312
EMORE,271,272,282,281
E,225,227,257,255,297,299,329,327
EMORE,226,242,256,241,298,314,328,313
EMORE,272,273,283,282
E,227,229,259,257,299,301,331,329
EMORE,228,243,258,242,300,315,330,314
EMORE,273,274,284,283
E,229,231,261,259,301,303,333,331
EMORE,230,244,260,243,302,316,332,315
EMORE,274,275,285,284
E,231,233,263,261,303,305,335,333
EMORE,232,245,262,244,304,317,334,316
EMORE,275,276,286,285
E,233,235,265,263,305,307,337,335
EMORE,234,246,264,245,306,318,336,317
EMORE,276,277,287,286
E,235,237,267,265,307,309,339,337
EMORE,236,247,266,246,308,319,338,318
EMORE,277,278,288,287
TYPE,2
MAT,300
R,1,0.000001
E,1,3,33,33,2,22,33,32
E,3,5,35,33,4,23,34,22
E,5,7,37,35,6,24,36,23
E,7,9,39,37,8,25,38,24
E,9,11,41,39,10,26,40,25
E,11,13,43,41,12,27,42,26
E,13,15,45,43,14,28,44,27
E,15,17,47,45,16,29,46,28
E,17,19,49,47,18,30,48,29
E,19,21,51,49,20,31,50,30
WSTART,1,1,2
WMORE,2,2,20
WMORE,32,32
WMORE,32,32,1
WSTART,3,3,2,2,30
WMORE,4,4,30
WMORE,22,22,1
WSTART,5,5,2,2,30
```

```
WMORE,6,6,30
WMORE,23,23,1
WSTART,7,7,2,2,30
WMORE,8,8,30
WMORE,24,24,1
WSTART,9,9,2,2,30
WMORE,10,10,30
WMORE,25,25,1
WSTART,11,11,2,2,30
WMORE,12,12,30
WMORE,26,26,1
WSTART,13,13,2,2,30
WMORE,14,14,30
WMORE,27,27,1
WSTART,15,15,2,2,30
WMORE,16,16,30
WMORE,28,28,1
WSTART,17,17,2,2,30
WMORE,18,18,30
WMORE,29,29,1
WSTART,19,19,2,2,30
WMORE,20,20,30
WMORE,30,30,1
WAVES
NLIST,ALL
ELIST,ALL
/VIEW,,-1,2,-1 $ENUM,1 $EPLOT
cp,1,uz,2,32
cp,2,uz,3,22,33
CP,4,UZ,4,34
CP,5,UZ,5,23,35
CP,7,UZ,6,36
CP,8,UZ,7,24,37
CP,10,UZ,8,38
CP,11,UZ,9,25,39
CP,13,UZ,10,40
CP,14,UZ,11,26,41
CP,16,UZ,12,42
CP,17,UZ,13,27,43
CP,19,UZ,14,44
CP,20,UZ,15,28,45
CP,22,UZ,16,46
CP,23,UZ,17,29,47
CP,25,UZ,18,48
CP,26,UZ,20,50
CP,27,UZ,21,31,51
cp,29,uX,2,32
cp,30,uX,3,22,33
CP,32,UX,4,34
CP,33,UX,5,23,35
CP,35,UX,6,36
CP,36,UX,7,24,37
CP,38,UX,8,38
CP,39,UX,9,25,39
CP,41,UX,10,40
CP,42,UX,11,26,41
```

CP,44,UX,12,42
CP,45,UX,13,27,43
CP,47,UX,14,44
CP,48,UX,15,28,45
CP,50,UX,16,46
CP,51,UX,17,29,47
CP,53,UX,18,48
CP,54,UX,20,50
CP,55,UX,21,31,51
CP,56,UZ,74,104
CP,57,UZ,75,94,105
CP,58,UZ,76,106
CP,59,UZ,77,95,107
CP,60,UZ,78,108
CP,61,UZ,79,96,109
CP,62,UZ,80,110
CP,63,UZ,81,97,111
CP,64,UZ,82,112
CP,65,UZ,83,98,113
CP,66,UZ,84,114
CP,67,UZ,85,99,115
CP,68,UZ,86,116
CP,69,UZ,87,100,117
CP,70,UZ,88,118
CP,71,UZ,89,101,119
CP,72,UZ,90,120
CP,73,UZ,91,102,121
CP,74,UZ,92,122
CP,75,UZ,93,103,123
CP,76,UX,74,104
CP,77,UX,75,94,105
CP,78,UX,76,106
CP,79,UX,77,95,107
CP,80,UX,78,108
CP,81,UX,79,96,109
CP,82,UX,80,110
CP,83,UX,81,97,111
CP,84,UX,82,112
CP,85,UX,83,98,113
CP,86,UX,84,114
CP,87,UX,85,99,115
CP,88,UX,86,116
CP,89,UX,87,100,117
CP,90,UX,88,118
CP,91,UX,89,101,119
CP,92,UX,90,120
CP,93,UX,91,102,121
CP,94,UX,92,122
CP,95,UX,93,103,123
CP,96,UZ,146,176
CP,97,UZ,147,166,177
CP,98,UZ,148,178
CP,99,UZ,149,167,179
CP,100,UZ,150,180
CP,101,UZ,151,168,181
CP,102,UZ,152,182

CP,103,UZ,153,169,183
CP,104,UZ,154,184
CP,105,UZ,155,170,185
CP,106,UZ,156,186
CP,107,UZ,157,171,187
CP,108,UZ,158,188
CP,109,UZ,159,172,189
CP,110,UZ,160,190
CP,111,UZ,161,173,191
CP,112,UZ,162,192
CP,113,UZ,163,174,193
CP,114,UZ,164,194
CP,115,UZ,165,175,195
CP,116,UX,146,176
CP,117,UX,147,166,177
CP,118,UX,148,178
CP,119,UX,149,167,179
CP,120,UX,150,180
CP,121,UX,151,168,181
CP,122,UX,152,182
CP,123,UX,153,169,183
CP,124,UX,154,184
CP,125,UX,155,170,185
CP,126,UX,156,186
CP,127,UX,157,171,187
CP,128,UX,158,188
CP,129,UX,159,172,189
CP,130,UX,160,190
CP,131,UX,161,173,191
CP,132,UX,162,192
CP,133,UX,163,174,193
CP,134,UX,164,194
CP,135,UX,165,175,195
CP,136,UZ,218,248
CP,137,UZ,219,238,249
CP,138,UZ,220,250
CP,139,UZ,221,239,251
CP,140,UZ,222,252
CP,141,UZ,223,240,253
CP,142,UZ,224,254
CP,143,UZ,225,241,255
CP,144,UZ,226,256
CP,145,UZ,227,242,257
CP,146,UZ,228,258
CP,147,UZ,229,243,259
CP,148,UZ,230,260
CP,149,UZ,231,244,261
CP,150,UZ,232,262
CP,151,UZ,233,245,263
CP,152,UZ,234,264
CP,153,UZ,235,246,265
CP,154,UZ,236,266
CP,155,UZ,237,247,267
CP,156,UX,218,248
CP,157,UX,219,238,249
CP,158,UX,220,250

```
CP,159,UX,221,239,251
CP,160,UX,222,252
CP,161,UX,223,240,253
CP,162,UX,224,254
CP,163,UX,225,241,255
CP,164,UX,226,256
CP,165,UX,227,242,257
CP,166,UX,228,258
CP,167,UX,229,243,259
CP,168,UX,230,260
CP,169,UX,231,244,261
CP,170,UX,232,262
CP,171,UX,233,245,263
CP,172,UX,234,264
CP,173,UX,235,246,265
CP,174,UX,236,266
CP,175,UX,237,247,267
CPLIST,ALL
D,ALL,UY,,,,,ROTX,ROTY,ROTZ
D,1,UX
D,52,UX
D,73,UX
D,124,UX
D,145,UX
D,196,UX
D,217,UX
D,268,UX
D,289,UX
D,19,UY
D,19,UZ
D,30,UZ
D,49,UZ
EP,31,5,220
EP,32,6,220
EP,33,6,220
EP,34,6,220
EP,35,6,220
EP,36,6,220
EP,37,6,220
EP,38,6,220
EP,39,6,220
FINISH
/solution
outpr,basic,1
solve
finish
```