NASA Contract Report CR-182235

The MHOST Finite Element Program:

3-D Inelastic Analysis Methods for Hot Section Components

Volume II - Users' Manual

Shohei Nakazawa
The Finite Element Factory
Sunnyvale, California

July 1989

Prepared for
Lewis Research Center
Under Contract NAS3 - 23697

National Aeronautics and
Space Administration

(NASA-CR-182235-Vol-2) THE MHOST FINITE ELEMENT PROGRAM: 3-D INELASTIC ANALYSIS METHODS FOR HOT SECTION COMPONENTS. VOLUME 2: USER'S MANUAL Final Report (MARC Analysis Research Corp.) 135 p CSCL 20K G3/39 N90-10450 Unclas 0217540

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
CR-182235		

4. Title and Subtitle	5. Report Date
The MHOST Finite Element Program: 3-D Inelastic Analysis Methods for Hot Section Components Volume II - Users' Manual	July, 1989
	6. Performing Organization Code: 533-04-11
7. Author(s): Shohei Nakazawa	8. Performing Organization Report No.: FEF/PD/1101/89
	10. Work Unit No.
9. Performing Organization Name and Address: MARC Analysis Research Corporation, 260 Sheridan Avenue, Palo Alto, CA 94306; The Finite Element Factory, 1767 Wright Avenue, Sunnyvale, CA 94087	11. Contract or Grant No.: NAS3-23697
	13. Type of Report and Period Covered: Final Report
12. Sponsoring Agency Name and Address: National Aeronautics and Space Administration, Lewis Research Center; Project Manager, D. A. Hopkins, MS49-8; 21000 Brookpark Road, Cleveland, OH 44135	14. Sponsoring Agency Code: RTOP 533-04-1A

15. Supplementary Notes

CR-182205 The MHOST Finite Element Program Volume I - Theoretical Manual

CR-182236 The MHOST Finite Element Program Volume III - Systems Manual

16. Abstract

This document describes the user options available for running the MHOST finite element analysis package. MHOST is a solid and structural analysis program based on the mixed finite element technology, and is specifically designed for three-dimensional inelastic anasysis. A family of two- and three- dimensional continuum elements along with beam and shell structural elements can be utilized. many options are available in the constitutive equation library, the solution algorithms and the analysis capabilities.

This document discusses the outline of solution algorithms, data input and output, analysis options including the user subroutines and the definition of the finite elements implemented in this program package.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
3-D Inelastic Analysis, Finite Elements, High Temperature, Creep, Vibration, Buckling, Solution Methods, Constitutive Modeling	General Release

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price*
Unclassified	Unclassified		

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

MHOST VERSION 4.2 USERS MANUAL
PREFACE

This manual describes the user options available for running the MHOST finite element analysis package. MHOST is a solid and structural analysis program based on mixed finite element technology, and is specifically designed for three-dimensional inelastic analyses. A family of two- and three-dimensional continuum elements along with beam and shell structural elements can be utilized. Many options are available in the constitutive equation library, the solution algorithms and the analysis capabilities.

An overview of the solution algorithms is presented in Section A. A general description of the input data formats is given in Section B, and input data for selecting solution algorithms are explained in Section C. The model definition data anf the incremental data discussed in sections D and E respectively represent the bulk of the input data required for a given analysis.

Example problems illustrating MHOST input and output are provided in Volume II Demonstration Problems. Most of the examples are simple but fully worked out. It is advisable for a new MHOST user to run the same examples problems. Then the new user can start exploring and exploiting the MHOST code to solve his/her own problems.

An unusually high level of flexibility is achieved in the MHOST code by allowing the user to prepare his/her own subroutines. As currently coded, user subroutines may only be used to enhance the mechanics theory in the code. A more detailed explanation of user subroutines is presented in Section F.

Finite elements available in the current version of the MHOST code are summarized in Section G. This section can be referred to by users for quick reference when constructing a finite element model.

MHOST output files are described in Sections H, I, and J. Because the MHOST program utilizes mixed finite element methods, all information calculated in the code is represented at nodes. The effect of this upon the interpretation of output and graphics post-processing is discussed in Sections H, I and J.

ACKNOWLEDGEMENT

The first version of this document was prepared by J. C. Nagtegaal. In the process of developing and enhancing this document, contributions are also made by J. B. Dias and M. S. Spigel. Earlier versions of the typescripts was prepared by P. Bartholow and D. Puccini. The author is responsible for the final manuscript.

PROGRAM MHOST USERS' MANUAL
Version 4.2

TABLE OF CONTENTS

PAGE

A. INTRODUCTION.. 1

B. DATA INPUT.. 9

B.1 Input Structure.. 9
B.2 Line Formats... 10
B.3 Numerical Data... 10
B.4 Continuation Lines... 11
B.5 Files.. 11
B.6 Programming Considerations................................... 12

C. PARAMETER DATA.. 15

D. MODEL DATA.. 34

E. INCREMENTAL DATA INPUT... 56

F. USER SUBROUTINES.. 62

G. ELEMENT LIBRARY... 79

Four-Node Plane Stress Element
- MHOST Element 3.. 81
Eight-Node Three-Dimensional Solid Element
- MHOST Element 7.. 85
Four-Node Axisymmetric Solid Element
- MHOST Element 10... 89
Four-Node Plane Strain Element
- MHOST Element 11... 93
Four-Node Shell Element
- MHOST Element 75... 97
Two-Node Beam Element
- MHOST Element 98... 103
Four-Node Assumed Strain Plane Stress Element
- MHOST Element Type 151.. 107
Four-Node Assumed Strain Plane Strain Element
- MHOST Element Type 152.. 111
Four-Node Assumed Strain Axisymmetric Element
- MHOST Element Type 153.. 115
Eight-Node Assumed Strain Solid Element
- MHOST Element Type 154.. 119

TABLE OF CONTENTS (Cont'd)

		PAGE
H.	LINE PRINTER OUTPUT	123
I.	LOG FILE	125
J.	POST PROCESSING FILE	126
K.	MHOST INDEX	127

PROGRAM MHOST USERS' MANUAL
Version 4.2

LIST OF FIGURES

		PAGE
Figure A.1	Flow Diagram for the Iterative Solution Procedure......	7
Figure A.2	Nonlinear Finite Element Solution Control Structure....	8
Figure B.1	MHOST Finite Element Driver Routines...................	14
Figure D.1	Coordinate Transformations for Anisotropic Material Axes Orientation..	51
Figure G.1	Plane Stress Element Traction Loading Definition.......	84
Figure G.2	Solid Element Traction Loading Definition..............	88
Figure G.3	Axisymmetric Solid Element Traction Loading Definition.	92
Figure G.4	Plane Strain Element Traction Loading Definitions......	96
Figure G.5	Shell Element Traction Loading Definitions.............	102
Figure G.6	Beam Element Traction Loading Definitions..............	106
Figure G.7	Connectivity and Loading for Element Type 151..........	110
Figure G.8	Connectivity and Loading for Element Type 152..........	114
Figure G.9	Connectivity and Loading for Element Type 153..........	118
Figure G.10	Connectivity and Loading for Element Type 154..........	122

PROGRAM MHOST USERS' MANUAL
Version 4.2

LIST OF TABLES

		PAGE
Table A.1	MHOST Analysis Capability	2
Table G.1	Element Type Characterization Table	79

THE MHOST PROGRAM

A. INTRODUCTION

The MHOST program is a finite element code designed to perform nonlinear analysis of turbine engine hot section components. MHOST has been developed at MARC Analysis Research Corporation under NASA Lewis Research Center Contract NAS3-23697, as a subcontractor to the United Technologies Corporation -Pratt and Whitney Division.

The innovative mixed finite element procedures implemented in the MHOST program result in significantly improved accuracy. The procedures allow formulation and solution of a system of nonlinear governing equations for inelastic structures, based upon concepts of mixed variational principles. The standard displacement method is also included as an option.

The current version of the MHOST code contains three types of inelastic constitutive models:

(i) simplified plasticity,
(ii) conventional plasticity, and
(iii) an advanced viscoplasticity law.

Temperature effects and material anisotropy can be obtained through user subroutines. The program contains routines for eigenvalue extraction in buckling and modal analyses. A single step time integrator routine is available for use in nonlinear transient calculations. Additional capabilities can be obtained by adding user subroutines. All the elements available in the current version are numerically integrated linear or quadratic isoparametric elements. All quantities, such as stresses and strains, input or internally generated, are represented by nodal points values.

The user interface for the MHOST code and its data processing is somewhat similar to that for a conventional finite element package such as the MARC General Purpose Finite Element code. A few additional parameters are required here to control the iterative procedure, as well as the numerical integration of various vectors associated with the nodal stress/ strain calculations.

In this document, data preparation and interpretation of computed results are discussed. A brief note on the underlying theory and programming is also included. For further theoretical details, users should refer to the MHOST Theoretical Manual and the listing of the code in the MHOST Systems Manual.

The MHOST control structure allows the user to extract eigenvalues and eigenvectors, after a quasi-static stress analysis, for either normal mode or buckling load calculations. The analysis capabilities and input/output options are given in Table A.1.

TABLE A.1 MHOST Analysis Capability

Element Definition Options	Beam	Plane Stress	Plane Strain	Axi-symmetric Solid	Three-Dimensional Solid	Three-Dimensional Shell
Linear Isotropic Elasticity	x	x	x	x	x	x
Anisotropic[*1] Elasticity		x	x	x	x	
Composite Laminate						x
Simplified Plasticity		x	x	x	x	x
Elasto-Plasticity		x	x	x	x	x
Unified Creep-Plasticity		x	x	x	x	x
Stress Stiffening	x	x	x	x	x	x
Centrifugal Mass	x	x	x	x	x	x
Thermal[*2] Strain	x	x	x	x	x	x
Creep Strain	x	x	x	x	x	x

NOTES:

[*1]Applicable only for linear elasticity.

[*2]Not separately applicable to the unified creep-plasticity option in which these quantities are combined together in the model.

TABLE A.1 MHOST Analysis Capability (Cont'd)

Analysis Module Option	Beam	Plane Stress	Plane Strain	Axi-symmetric Solid	Three-Dimensional Solid	Three-Dimensional Shell
Quasi-static Analysis *3	x	x	x	x	x	x
Buckling Analysis *3	x	x	x	x	x	x
Modal Analysis *3	x	x	x	x	x	x
Modal Superposition *3	x	x	x	x	x	x
Transient Dynamics	x	x	x	x	x	x
Large Displacement Analysis	x	x	x	x	x	x

Notes:

*3 The effect of stress stiffening and/or centrifugal mass may be included.

TABLE A.1 MHOST Analysis Capability (Cont'd)

Iterative Solution Option	Quasi-static Profile Solution	Quasi-static Frontal Solution	Transient Profile Solution	Transient Frontal Solution
Modified[*4] Newton	x	x	x	
Newton Raphson Update	x	x	x	
Conjugate[*5] Gradient	x	x	x	
BFGS Update	x	x	x	
Secant Newton Update	x	x	x	
Line Search	x	x	x	

Notes:

[*4]Updating of the tangent array can be stopped at an arbitrary step of the Newton-Raphson stiffness update process

[*5]The line search option is automatically turned on.

The MHOST program is written in FORTRAN, with the ANSI-68 standard followed as closely as possible. All floating point calculations are carried out in double-precision (64 bits/word) on the 32 bits/real-word machines (e.g., PRIME, VAX and IBM), or in single-precision (60-64 bits/word) on the higher precision machines (such as CDC and CRAY).

INCREMENTAL-ITERATIVE SOLUTION PROCEDURE

This section provides a brief discussion of the numerical procedure used in the MHOST program. It is not our intention here to discuss the procedure in detail, but rather to provide a user with sufficient background to enable him/her to use the MHOST program efficiently for inelastic analyses. A comprehensive discussion of the solution procedures used in the MHOST code is contained in the MHOST Theoretical Manual.

The iterative finite element solution procedure is illustrated by a flow diagram as shown in Figure A.1. Matrices K and F are the conventional finite element stiffness matrix and load vector, respectively. Other conventional finite element matrices appearing in this flow diagram are B, D, and Q, respectively the strain-displacement matrix, the stress-strain matrix and the diagonalized inner product of shape functions. All quantities involved in the computation are evaluated as nodal vectors. Values for stresses, strains and displacements are obtained at nodes by the nodal strain recovery calculation indicated in Box 3 and the residual load correction in Boxes 2, 4 and 5 of the flow diagram. Values at integration points are calculated and printed only when requested and these results can be compared with those of the conventional finite element method.

Iterative solutions are utilized even when solving for purely elastic response. An iterative solution of elastic response, compared to the standard displacement method, produces improved stress results for the same accuracy in displacement field. In path-dependent inelastic calculations, such improved response takes on larger importance.

The user can try various options to conduct an iterative calculation procedure. Options include the different criteria to terminate the iterative calculation as well as numerical quadrature rules to form various matrices and vectors. Input data related to the iterative solution are documented in Section C.10, for the numerical integration, and D.6 for termination of the process. Various additional options have been added to the code to improve computational efficiency. It is generally recommended that a user test the options to be able to choose the most efficient combination for his/her particular problem.

The iterative solution procedure is embedded in the conventional incremental finite element control structure as shown in Figure A2. In the incremental calculation, the tangent stiffness matrix is formed at each increment (Full-Newton). Within each increment, the iterative algorithm performs in the same manner as the Newton-Raphson procedure for solving a system of nonlinear equations. The Full-Newton procedure is the default in MHOST with the modified-, quasi- and secant-Newton iterations as options. A self- adaptive load incrementation option based upon the arc-length method can be invoked for computation of the tangent stiffness. When an optional method is invoked, the operation in Box 1 of Figure 2 is skipped after the specified number of iterations.

Section E of this document describes the input data associated with the incremental procedure used for nonlinear and transient analyses.

MHOST provides the default values for parameters associated with the integration and the iterations, however the user is encouraged to overwrite the option defaults to explore the possibilities of the novel computational technology implemented into the MHOST code.

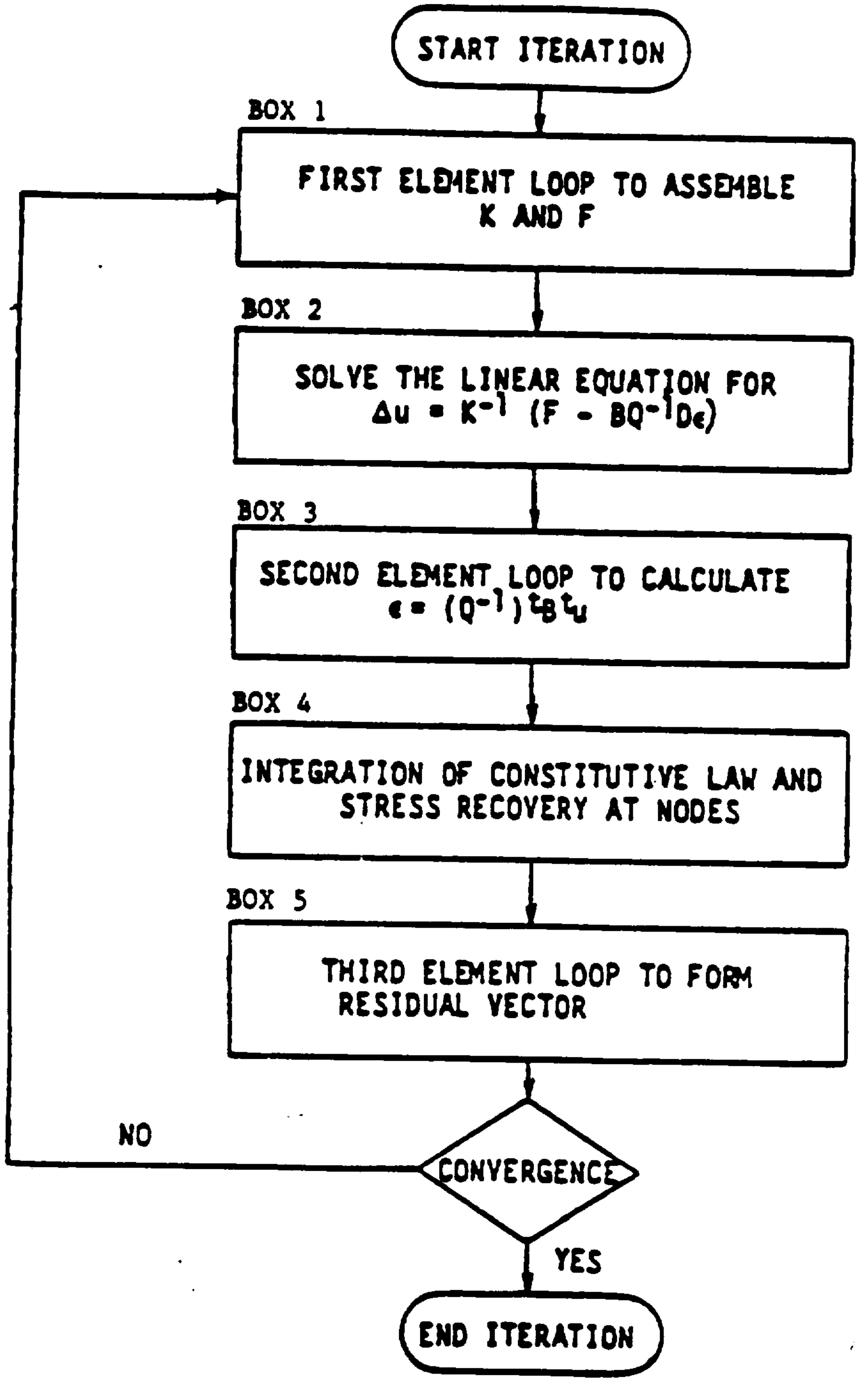

FIGURE A.1 Flow Diagram for Iterative Solution Procedure

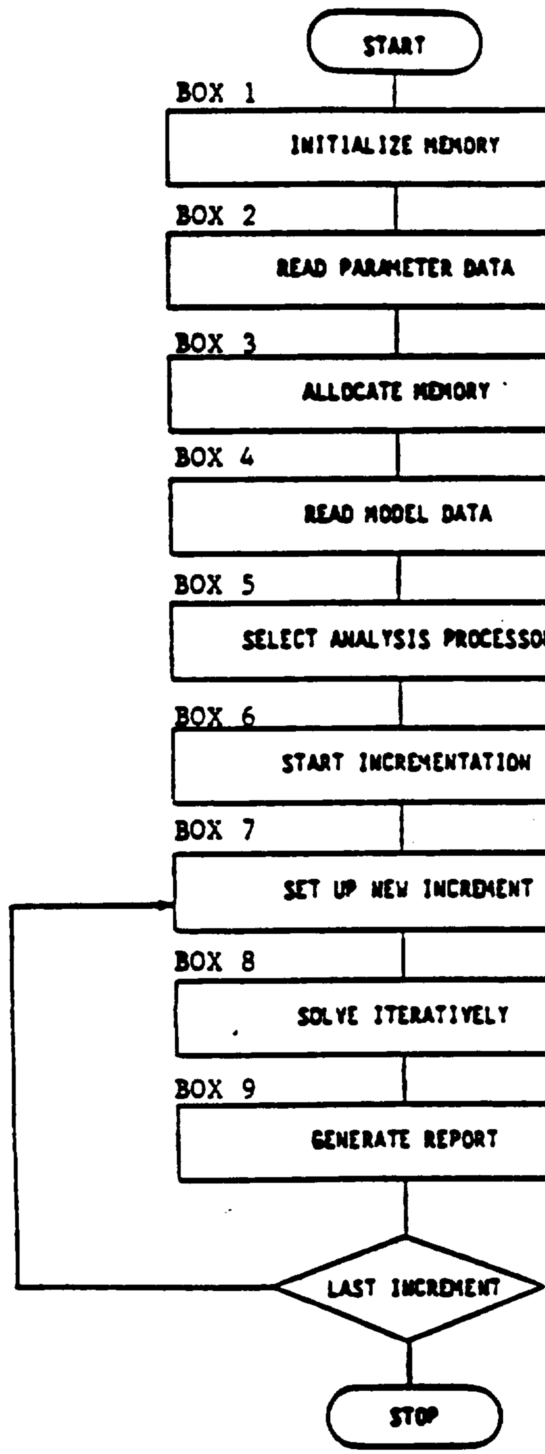

FIGURE A.2 Nonlinear Finite Element Solution Control Structure

B. DATA INPUT (General)

B.1 Input Structure

The data input to the program MHOST consists of three blocks.

1. The PARAMETER DATA block serves to specify and control MHOST execution options (e.g., modal analysis, stress stiffening, etc.). Also, in this block, upper bounds are specified for various model parameters (e.g., the number of elements, nodes, etc.) to permit the dynamic memory array allocations to be set.

2. The finite element model is defined with the MODEL DATA block. In this block of data finite element topology, nodal coordinates, boundary conditions and loads are specified.

3. The INCREMENTAL DATA block specifies the loading history as well as modifications to initial and boundary conditions. MHOST has no limit on the number of incremental data blocks.

The initial conditions and periodic loading conditions for the transient dynamic calculations are specified as part of the first INCREMENTAL DATA block.

Each block is terminated with a line containing the keyword END. All three blocks will contain a number of SEGMENTS. Each segment consists of a line with a KEYWORD, and if necessary followed by a number of lines with data. Segments can be entered in an arbitrary sequence within a block. Each segment type may appear more than once in the same block. During the second and subsequent appearances of a segment in a block, the data entered is processed without taking into account previously specified data input. In many cases this will mean that the previously specified data is erased. In some cases, it is also possible to complement data specified earlier.

The MHOST input processor only requires the first four characters of a keyword. Therefore, any KEYWORD can be abbreviated by its first four characters.

When initial conditions are specified for dynamic transient computations, a block of data segments must be added to the data deck as a part of the first incremental data block. A special key word to identify this block is required. History dependent loading data must be specified at each time increment unless otherwise given by means of user subroutines.

The MHOST input data processor reads all of the input data character-by-character using FORTRAN A1 format. The input data file is processed completely with checks made for possible typing errors and data inconsistency. When such a mistake is detected, the program prints comprehensible error messages and terminates execution after processing the remaining input data.

B.2 Line Formats

In both keywords and numerical data input, items are separated from each other by one or more blank spaces. There is no limitation as to where data may start or end on a line. The end of a line is also considered a separator; hence, every data item must be included in its entirety on one line.

MHOST accepts four types of input lines:

a. KEYWORD lines

These lines always start with an asterisk (*), followed by the keyword. The star may be preceded by blank, and blanks may also be included between the start and the keyword. The keyword may be followed by several numbers, separated from the keyword and each other by blanks. A keyword is recognized by its first four characters.

b. COMMENT lines

Comment lines may be included by the user to explain various parts of his input. MHOST comment lines start with a C, optionally preceded and/or followed by blanks. The text on a comment line is reflected literally in the output file. Otherwise comment lines have no influence. Completely blank lines are treated as comment lines.

c. DATA lines

These lines contain only numbers. DATA lines cannot start with a * or a C. Numbers are separated from each other by blanks.

d. The TITLE line

The first line of the input file is considered to represent the title of the problem. This line is printed at the top of every page.

B.3 Numerical Data

Numerical input data consists of integer and real data.

An INTEGER consists of a number of digits, optionally preceded by a plus (+) or a minus (-) sign. Blanks are not allowed within the number.

A REAL can be specified in various ways. The simplest way is to use the same format as used for integer specification. Alternatively, a REAL can be specified with a decimal point. The decimal point can be positioned anywhere within the range of digits, or before or after the complete series of digits, but never before the plus (+) or minus (-) sign, if present.

If the REAL consists solely of a decimal point (optionally preceded by a sign), the REAL has the value zero. This means that '.', '+'. and '-'. all will be read as zero.

Finally, a REAL can be specified with an exponent. The exponent consists of the letter E followed by an integer, and is attached at the end of a real as specified above. If only the exponent is specified, it is assumed that it is preceded by a real of value 1 (i.e. E - 10 = 1.0E-10). If a line contains more numeric entries than required, the remaining numeric entries will not be read in. In addition, it is possible to exclude the E in the exponent if the value of the exponent is preceded with either a + or - (i.e., 2.3 +6 = 2.3E6). If a line contains fewer numeric entries than required, the remaining numeric entries will be set equal to zero.

B.4 Continuation Lines

The number of integers and reals on a line is tied to certain maxima: no more that 16 integers or 8 reals can be entered on a single line. If both integers and reals appear on a line, the limit can be formulated as:

$$(\text{No. of integers})*5 + (\text{No. of reals})*10 < 80.$$

If an input record exceeds the above limit, continuation lines are used, which have the same specification as regular numerical data lines. Continuation lines are only used following numerical data lines in which no space for further data is provided.

B.5 Files

The MHOST program uses six files for input and output. The files and their contents are listed below:

Unit	Contents
1	Log file
5	Input data
6	Input Echo, Error messages, Iteration data, Processing Information and Results.
8	Restart data. (binary form)
11	Scratch file.
19	Post-processing file

The names of the above files are system dependent.

B.6 Programming Considerations

The parameter data lines in the input file are read by SUBROUTINE DATIN1. SUBROUTINE DATIN1 sets up flags and parameters used internally to control execution.

MHOST utilizes dynamic core allocation for memory management. Core allocation is carried out in SUBROUTINE INITI1 which is entered from the model data reader SUBROUTINE BULKIN. SUBROUTINE INITI1 gives values to pointers for the mesh data storage. Subsequently, SUBROUTINE DATIN2 is entered to read the model data. Individual data input modules are executed with respect to the key words.

Most trivial input data errors are detected by the code itself. MHOST also performs a cross-reference check of the parameter and model data for consistency.

It is advisable for a user to go through the input data whenever something goes wrong. For the post-mortem, the MHOST Systems Manual should be consulted.

Once the input of the model data has been completed, one of the finite element driver routines is entered. These routines are listed in Figure B.1. The coding strategy is that a new analysis capability is added to the system by introducing a new driver routine. Each driver routine uses common finite element and constitutive equation libraries.

The incremental data input is processed by SUBROUTINE INCRIN which calls SUBROUTINES DATIN3 and DATOH1. In SUBROUTINE DATIN3, the incremental data lines are read and added to the existing in-core database. The automatic and adaptive load incrementations are processed in this subroutine. SUBROUTINE DATOH1 generates the report on the incremental update of loading data.

The finite element solution is printed by SUBROUTINE PRINOU which is used by are driver routines.

When requested, the post-file generator SUBROUTINE POSTOU is entered and a formatted file is written for graphics post-processing. The format is consistent with the MARC general purpose finite element package and can be processed by MENTAT* with a MHOST enhancement.

*MENTAT is a proprietary graphics/finite element modeling software package available from MARC Analysis Research Corporation.

The finite element driver subroutines execute the finite element analysis modules. The execution is terminated by a driver routine only when the iterative process fails to converge within the specified number of iterations. The execution time of each operation is reported in the line printer file (FORTRAN unit number 21).

Major analysis modules are SUBROUTINE ASSEM1 which constructs the element stiffness matrices, and assembles the global stiffness matrix and load vector. SUBROUTINE ASSEM2 performs frontal (wave-front) solution of equations. SUBROUTION ASSEM3 is used to generate the coefficient matrices for transient time integration. Nodal strain and stress arrays are computed in SUBROUTINE ASSEM4. No diagnostics are generated in these routines.

The above finite element analysis modules control the subroutines which contain both element and constitutive equation libraries. SUBROUTINE DERIV controls the element library at the element level. SUBROUTINE CNODEL is used in the element loop to construct element arrays. SUBROUTINE NODSTR is called by SUBROUTINE STRESS and supervises the execution of element library routines. SUBROUTINE NODSTR controls the loop over the nodal points and, for shell elements, the integration layers in conjunction with the pre-integration.

The subelement iteration loop is entered from the residual force vector loop in SUBROUTINE ASSEM4. The main control routine is SUBROUTINE SUBFEM, which calls the subroutines needed for a complete mixed iterative solution in a subelement region. Execution timing is reported in the subelement solution process. For the subelement solution, the element library subroutines are constructed slightly different from the global solution. The constitutive equation library is accessed via SUBROUTINE STRESS in exactly the same manner as the global solution.

MAIN Program - BLOCK DATA SUBPROGRAM

SUBROUTINE HOST

- SUBROUTINE DATIN1
  Reads parameter data

- SUBROUTINE BULKIN
  Allocates core, reads model data and echoes the input data

- SUBROUTINE INCRIN
  Reads the incremental data and echoes the input data. This routine is called from the analysis driver routines, inside of the incremental solution loop.

- SUBROUTINE STATIC
  Drives the incremental iterative solution for quasi-static analysis (with the band matrix option)

- SUBROUTINE DYNAMT
  Drives the direct time integration for transient analysis

- SUBROUTINE MODAL
  Drives the eigenvalue extraction subsystem for modal analysis of linear elastic structures

- SUBROUTINE SUPER
  Drives modal superposition for linear elastic structures. Mode shapes and frequencies are obtained from SUBROUTINE MODAL.

- SUBROUTINE FREDOM
  Drives frequency domain analysis for linear elastic structures.

- SUBROUTINE FRONTS
  Drives the incremental iterative solution for the quasi-static analysis (with the frontal solution option)

- SUBROUTINE BUCKLE
  Drives the linear buckling analysis.

- SUBROUTINE PRINOU
- SUBROUTINE POSTOU

Generate reports on the line printer file and the formatted post-processing file. These routines are called from the analysis driver routines.

FIGURE B.1 MHOST Finite Element Driver Routines

C. PARAMETER DATA

The parameter data immediately follow the title line. Each parameter segment consists generally of a single keyword line, containing the keyword and up to 4 integer parameter values. Only the segment *ELEMENTS contains additional numerical data lines. The description is given per segment, where the keyword is used as title for the segment.

The following 32 keywords can be used in Version 4.2:

SECTION C INDEX

KEYWORD	SECTION NO.	PAGE
*ANISOTROPY	C.15	22
*BANDMATRIX	C.35	26
*BASEEXCITATION	C.50	30
*BEAMSECTION	C.40	27
*BFGS	C.43	28
*BOUNDARY	C.3	19
*BUCKLE	C.17	22
*CENTRIFUGALMASS	C.53	30-31
*COMPOSITE	C.51	30
*CONJUGATEGRADIENT	C.54	31
*CONSTITUTIVE	C.19	23
*CREEP	C.14	22
*DASHPOTS	C.45	28-29
*DEFORMATION	C.37	26-27
*DISPLACEMENTMETHOD	C.41	27-28
*DISTRIBUTEDLOAD	C.20	23
*DUPLICATENODE	C.32	25
*DYNAMIC	C.12	21
*ELEMENTS	C.1	17
*EMBED	C.38	27
*END	C.58	33
*FINITE_STRAIN	C.61	33
*FOLLOWER_FORCE	C.62	33
*FORCES	C.6	19
*FRONTALSOLUTION	C.36	26
*HARDENING_SLOPE	C.59	32-33
*HARMONICLOAD	C.49	29-30
*LARGE_DISPLACEMENT	C.60	33
*LINESEARCH	C.47	29
*LOUBIGNAC	C.10	20
*MASSES	C.46	29
*MODAL	C.16	22
*NODES	C.2	18
*NOECHO	C.55	31
*OPTIMIZE	C.13	21
*PERIODICLOADING	C.31	25

*POST	C.33	25-26
*POWERSPECTRUM	C.56	31-32
*PRESSURE	C.58	32
*PRINTSETS	C.7	20
*PULSELOAD	C.57	32
*REPORT	C.21	23
*RESTART	C.8	20
*SCHEME	C.30	25
*SECANTNEWTON	C.48	29
*SHIFT	C.42	28
*SPRINGS	C.44	28
*STIFFENING	C.52	30
*STRESS	C.11	21
*TANGENT	C.22	24
*TEMPERATURE	C.9	20
*THERMAL	C.18	23
*TRANSFORMATIONS	C.5	19
*TYING	C.4	19
*UBOUN	C.34	26
*UCOEF	C.26	24
*UDERIV	C.29	25
*UFORCE	C.24	24
*UHOOK	C.28	25
*UPRESS	C.27	24
*UTEMP	C.25	24
*UTHERM	C.23	24
*WKSLP	C.39	27

C.1 *ELEMENTS

This parameter must be present to allocate memory for element connectivity. At least one element type data line is required.

WARNING: The execution will be terminated if this data line is not present.

Parameter 1: Upper bound to the number of elements.

The user should preferably specify the exact number of elements in the mesh as the parameter value. The core allocation and print line control are based on this parameter.

After the keyword line, a continuation line is used to define the element type to be used in the analysis. In the current version of MHOST finite element models are limited to a single element. One integer is to be specified on the following line, designating the element type.

The following element types are currently available:

3: 4-node plane stress element. Node and integration points are numbered counter-clockwise. Analogous to MARC element type 3. Five element properties: thickness, Young's modulus, Poisson's ratio, thermal expansion and density.

7: 8-node solid element. Node and integration points are numbered counter-clockwise. Analogous to MARC element type 7. Five element properties: dummy, Young's modulus, Poisson's ratio, thermal expansion and density.

10: 4-node axisymmetric solid element. Node and integration points are numbered counter-clockwise. First coordinate axis is symmetry axis. Analogous to MARC element type 10. Five element properties: dummy, Young's modulus, Poisson's ratio, thermal expansion and density.

11: 4-node plane strain element. Node and integration points are numbered counter-clockwise. Analogous to MARC element type 11. Five element properties: thickness, Young's modulus, Poisson's ratio, thermal expansion and density.

75: 4-node thick/thin shell element. Nodes and integration points are numbered counter-clockwise. Analogous to MARC element type 75. Five element properties: thickness, Young's modulus, Poisson's ratio, thermal expansion and density.

98: 2-node Timoshenko beam element. Analogous to MARC element type 98.

101: 9-node quadratic Lagrangian plane stress element. Node and integration points are numbered counterclockwise with the center node defined last. Other definitions are identical to those in element type 3.

102: 9-node quadratic Lagrangian plane strain element. Node and integration points are numbered counter-clockwise with the center node defined last. Other definitions are identical to those in element type 11.

103: 9-node quadratic Lagrangian axisymmetric element. Node and integration points are numbered counter-clockwise with the center node defined last. Other definitions are identical to those in element type 10.

151: 4-node plane stress element based on an assumed strain formulation. Node and integration points are numbered counterclockwise. Six entries for element properties include the thickness, Young's modulus, Poisson's ratio, thermal expansion coefficient, mass density and (optional) shear modulus, in that order.

152: 4-node plane strain element based on an assumed strain formulation. Node and integration points are numbered counterclockwise. Six entries for element properties include a dummy parameter, Young's modulus, Poisson's ratio, thermal expansion coefficient, mass density and (optional) shear modulus, in that order.

153: 4-node axisymmetric element based on an assumed strain formulation. Node and integration points are numbered counterclockwise. Five entries for element properties include a dummy parameter, Young's modulus, Poisson's ratio, thermal expansion coefficient and mass density, in that order. The shear modulus should not be specified separately, since this would make little sense for an axisymmetric problem!

154: 8-node solid element based on an assumed strain formulation. Node and integration points are numbered in accordance with the convention shown in Figure G.10. Six entries for element properties include a dummy parameter, Young's modulus, Poisson's ratio, thermal expansion coefficient, mass density and (optional) shear modulus, in that order.

Detailed element descriptions are provided in Section G.

C.2 *NODES

This keyword must be present to allocate memory for the nodal point coordinate values. The number of entries at each nodal point is calculated in the code.

It is recommended that the user specify the exact number of nodal points in the mesh. Core allocation and print line control are based on this number.

Parameter 1: Upper bound to the number of nodes.

C.3 *BOUNDARY

This keyword allocates memory for nodal displacement constraint data. Except for a few special cases, at least the same number of boundary conditions as the rigid body modes must be specified.

Insufficient specification of the boundary conditions can result in a singular stiffness matrix.

Parameter 1: Upper bound to the number of boundary conditions.

C.4 *TYING

This keyword allocates memory for constraint equations between nodal degrees-of-freedom. The coefficients and locations of these equations are specified by a data segment accompanied by the same keyword in the model data.

No constraint is imposed if the corresponding data segment is provided in the model data section. No diagnostics are generated.

Parameter 1: Upper bound to the number of tying equations

Parameter 2: Upper bound to the number of degrees-of-freedom involved in a tying equation.

C.5 *TRANSFORMATIONS

This keyword allocates memory for coordinate transformations of nodal degrees-of-freedom. The angles of rotation are specified by a data segment accompanied by the same keyword in the model data.

Transformations take place only when the data is supplied in the model data section. No diagnostics are generated.

Parameter 1: Upper bound to the number of transformations.

C.6 *FORCES

This keyword allocates memory for concentrated nodal forces. The same keyword is used to define the data segment for nodal force input in the model data and incremental data sections.

Nodal forces are not applied unless the data segments appear in the model data and incremental data sections.

Parameter 1: Upper bound to the number of nodal degrees-of-freedom having non-zero concentrated forces.

C.7 *PRINTSETS

This optional keyword does not need to be specified unless more than 10 print options are to be chosen, or if the iterative updates for incremental displacement and reaction forces are to be printed.

Parameter 1: Maximum number of sets to be used in print specification. Default is 10 sets.

A negative parameter value invokes the printing of iterative displacement updates.

C.8 *RESTART

If this line is included, a previous analysis is to be continued. No other parameter or model input will be used or is necessary in a restart input file. Only the *END line must be included. The binary restart file must be available on a file assigned to unit 8.

C.9 *TEMPERATURE

No parameters. Indicates that nodal temperatures can be stored and used in the calculations.

C.10 *LOUBIGNAC

This keyword invokes the mixed finite element solution algorithms and selects the integration schemes for stiffness, strain recovery and residual terms. With these options the type of mixed interpolation procedure may be controlled.

The following parameters are effective to the linear two and three-dimensional elements in MHOST Version 4.2.

For reduced integration, strain sampling and numerical quadrature are carried out at the centroid of the linear elements. For full integration, strain sampling and quadrature are carried out at two points in each element coordinate direction.

For the trapezoidal, rule strain sampling and numerical quadrature are carried out at each corner of the quadrilateral element.

For selective integration the shear strain stiffness component is evaluated at the element centroid, with other strain components evaluated either by the full or trapezoidal integration rules.

The trapezoidal rule automatically implies use of selective integration in the present version of MHOST.

Parameter 1: 1. Reduced integration for strain recovery

2. Full integration for strain recovery

3. (Default) Trapezoidal rule for strain recovery

Parameter 2: 1. (Default) Full integration for residual force calculation

2. Reduced integration for residual force calculation

Parameter 3: 1. Full integration for stiffness matrix assembly.

2. Selective integration for stiffness matrix assembly.

3. (Default) Selective integration with element Cartesian coordinate transformation for stiffness matrix assembly.

*LOUBIGNAC 3 1 3 is the system default and recommended for most of the applications.

C.11 *STRESS

This option is used to allocate the memory for storing the nodal constraint for the stress components. It is not recommended to use this option in practical computations. When this option is invoked, erroneous results or divergence may occur.

Parameter 1: Upper bound to the number of nodal stress components that are to be prescribed.

C.12 *DYNAMIC

This option invokes the direct time integration procedure.

C.13 *OPTIMIZE

This options is used to flag the bandwidth optimization procedure.

Parameter 1: Number of iterations in the Cuthill-McKee band width optimization algorithm. The default value is 10.

When this option is invoked, the bandwidth is reported on both line printer and log files. Note that the bandwidth is calculated based on the node numbers and assuming a single degree-of-freedom per node. The subsequent analysis is performed using the first smallest nodal numbering achieved during the iteration. The actual bandwidth reported by the solver printed on the line printer file is the actual number of non-zero entries in the banded/profile equations. The relation is:

(Actual Half-bandwidth) =
(half-bandwidth reported by the optimizer)
x (number of degree-of-freedom per node) - 1

C.14 *CREEP

This option is invoked if the conventional creep model is being used.

C.15 *ANISOTROPY

This option is used to flag anisotropic material behavior. Anisotropy may be included for elasticity, yield surface, and thermal strains. Material axes orientation must be defined in the model data.

C.16 *MODAL

This option is used to flag the eigenvalue extraction option. Currently the subspace iteration method is utilized.

Parameter 1: Number of modes to be extracted, defaults to one if not given.

Parameter 2: Number of subspace dimensions for the eigenvalue extraction.
(A sufficient dimension is reserved as a default)

Parameter 3: Increment number at which eigenvalue extraction is to be performed. (Default = 0).

If the results from the modal extraction are to be used for a linear dynamics analysis using modal superposition, a continuation line must follow, defining

Parameter 1: Number of dynamic modes to be used for mode superposition.

C.17 *BUCKLE

This option is used to flag the linear elastic buckling analysis.

Parameter 1: Number of buckling modes to be extracted.

Parameter 2: Number of subspace dimensions for the eigenvalue extraction
(A sufficient value is assigned as a default)

Parameter 3: Increment number at which eigenvalue extraction is to be performed. (Default = 0).

C.18 *THERMAL

This option is used to flag temperature dependent material properties. These properties are given via user subroutine UTEMP.

C.19 *CONSTITUTIVE

This option selects the constitutive relation to be used. Note that the default is von Mises plasticity as indicated below, not linear elasticity. For linear elastic analysis, the user needs to flag the correct option here so as to avoid unnecessary computations.

When the *COMPOSITE option is used for shell elements, all the request for the constitutive equations are ignored.

Parameter 1: Option Type

0. Linear elasticity.
1. Simplified plasticity (secant elasticity) model.
2. Conventional plasticity (default).
3. Walker's creep plasticity model

When either the secant plasticity or the conventional plasticity model is flagged, the limit of elastic response must be specified by the *WORKHARD data segment (see Section D.18). If such data is missing, the secant elasticity model will result in a singular stiffness matrix and the conventional plasticity model will assume an astronomical number (10^{50}) as the yield stress.

When the Walker's creep-plasticity model is flagged, the user must provide parameter values through user subroutine WALCON. An example of this subroutine is included in this document. Also the MHOST program is shipped with this example. See Section F.9 for additional information.

C.20 *DISTRIBUTEDLOAD

The distributed load option (including gravity, centrifugal and traction loading types) are flagged. No parameter option is needed.

C.21 *REPORT

This option controls the output of nodal solutions.

Parameter 1: Print interval

Default is nodal solutions specified by the *PRINTOPTION (Section D.14) to be printed at the end of every increment. When this option is flagged, the solution is printed only at the intervals specified by the positive integer parameter.

C.22 *TANGENT

This option invokes the modified Newton-Raphson iteration procedure.

Parameter 1: Iteration cycle at which updating of the tangent stiffness matrix is halted in each increment.

When this option is flagged, the tangent is updated until the specified iteration count is met, after which no further reassembly of the stiffness matrix occurs. If this option is not flagged, the tangent stiffness matrix is updated at every iteration cycle except for purely linear elastic problems.

C.23 *UTHERM

This option invokes the call to the user subroutine UTHERM. No parameter. When this option is flagged, the user subroutine must be updated and compiled prior to the execution. This option prevents the user subroutine from interfering with other problems.

C.24 *UFORCE

This option invokes the call to the user subroutine UFORCE. No parameter. The user subroutine must be in the load module when MHOST is executed with this option.

C.25 *UTEMP

This option invokes the call to the user subroutine UTEMP. No parameter. The user subroutine must be in the load module when MHOST is executed with this option.

C.26 *UCOEF

This option triggers the call to the user subroutine UCOEF. No parameter. The user subroutine must be in the load module when MHOST is executed with this option.

C.27 *UPRESS

This option flags the user subroutine call to UPRESS. No parameter. The user subroutine must be in the load module when MHOST is executed with this option.

C.28 *UHOOK

This option invokes the call to user subroutine UHOOK. No parameter. The user subroutine must be in the load module when MHOST is executed with this option.

C.29 *UDERIV

This option flags the call to user subroutine UDERIV. The user subroutine must be in the load module when MHOST is executed with this option.

C.30 *SCHEME

The time integration scheme is altered by invoking this option. After the keyword, a free format numeric data line is used to specify integration parameters. The second data line consists of three reals. Those are:

Real 1 Reserved for future extension (set 0.0)

Real 2 Parameter b in Newmark family of single step schemes

Real 3 Parameter g in Newmark family of single step schemes

Note that the default value for b and g are 0.25 and 0.5, respectively, resulting in the average acceleration scheme. For possible combination of these parameters, see Theoretical Manual.

C.31 *PERIODICLOADING

This option is used to invoke periodic nodal force loadings and/or displacement boundary conditions. Two integer parameters.

Parameter 1: Maximum number of displacement conditions.

Parameter 2: Maximum number of nodal force conditions.

The loadings and/or boundary conditions must be defined in the incremental data.

C.32 *DUPLICATENODE

The duplicated node option is invoked by entering this parameter card. At duplicated nodes the displacement degrees-of-freedom are degenerated to the master node. Strains and stresses are still evaluated separately at each node.

Parameter 1: Maximum number of the duplicated nodal points.

C.33 *POST

This option invokes the generation of post-processing file (formatted on FORTRAN unit number 19) for plotting. The contents and format of the post-processing file are discussed in Section J.

Parameter 1: Post file generation interval.

C.34 *UBOUN

This parameter is used to flag the call to user subroutine UBOUN. The user subroutine must be in the load module when MHOST is executed with this option.

C.35 *BANDMATRIX

The symmetric profile matrix solver is invoked by using this option. Note that this option is the default.

No parameter.

The bandwidth optimization option can be used. No out-of-core solution capability is supported in the current version of MHOST.

C.36 *FRONTALSOLUTION

The optional nonsymmetric frontal solution subsystem is entered by flagging this parameter. This is an out-of-core solution and utilizes a direct access device connected to FORTRAN unit number 14.

No parameter.

This option cannot be used with the BANDMATRIX option. No wavefront optimization option has yet implemented in MHOST.

This option cannot be invoked when the dynamic analysis or the eigenvalue extraction option is flagged. For certain cases when the parameter specification is contradictory, the program generates appropriate error messages. Otherwise, this option will either be ignored or results in execution errors.

This option is not operational on IBM systems.

C.37 *DEFORMATION

This option is used to invoke eigenvalue extraction for the displacement stiffness matrix. Two integer parameters can be specified.

Parameter 1: Load step at which the deformation modes are extracted.

Parameter 2: Number of eigenvalues and eigenvectors to be extracted.

If the parameter is not given, the program automatically extracts the longest eigenvalue at the undeformed initial state.

This option is extremely useful to examine the characteristics and quality of elements and meshes. See O. C. Zienkiewicz and S. Nakazawa (1982), "The Penalty Function Method and its Applications to the Numerical Solution of Boundary Value Problems", ASME AMD. Vol. 51, pp 157-179. The information obtained from this calculation is equivalent to the results of the patch test.

C.38 *EMBED

This keyword flags the execution of the subelement iteration option.

One integer parameter can be specified to choose the integration algorithm for the subelement stress.

Parameter 1: 0 For area average trapezoidal integration.

1 For nodal trapezoidal integration.

C.39 *WKSLP

This keyword targets the user subroutine WKSLP to supply stress-strain data for elastic-plastic constitutive behavior. See Section F.4 for further details.

C.40 *BEAMSECTION

This option is used to specify one or more sets of cross-sectional properties for elastic beam elements.

Parameter 1: Upper bound to the number of beam element cross-sectional property sets. Defaults to 1.

C.41 *DISPLACEMENTMETHOD

This option disables the mixed iterative solution algorithm. The conventional displacement method is invoked for linear elastic problems. Use of this procedure for inelastic computations is not recommended.

No parameter.

When this option is flagged, the stress at a quadrature point is calculated based on the displacement method and is used to generate the reaction force vector in the equilibrium iteration. The total residual force does not become precisely zero due to round off. Therefore, it is advisable to specify a fictitious relative residual tolerance to avoid the ERROR logo print out and discontinuation of the execution.

The strain and stress are recovered at nodes even when this option is used for the solution. The MHOST code does not support any option to directly recover strains and stresses at quadrature points.

C.42 *SHIFT

This option is used to invoke the power shift capability, allowing the specification of one or more frequency ranges for eigenvalue search in linear dynamic analyses.

Parameter 1: Upper bound to the number of power shifts (frequency ranges) specified for eigenvalue search.

C.43 *BFGS

The quasi-Newton, inverse BFGS update process is invoked by this parameter. One integer parameter is required:

Parameter 1: Number of trial vectors to be stored in-core for the BFGS update. The default value is 10.

The default algorithm for the iterative solution of the mixed system of equations is the Newton-Raphson tangent stiffness assembly with factorization at each inelastic iteration process. BFGS avoids these operations and directly updates the inverse of the tangent stiffness. This requires only a few vector multiplications along with back substitution.

This option is potentially far more economical than the default procedure. It is advisable to flag this option when the standard procedure is unacceptably slow or fails to converge.

The initial tangent modulus is used in this solution algorithm (often referred to as the KT-1 method). The *TANGENT option is ignored when the BFGS parameter is specified.

This option cannot be used with the secant Newton option. BFGS is applicable for quasi-static and direct integration dynamic problems.

C.44 *SPRINGS

This option allows additional stiffness terms at a node to be added directly into the stiffness matrix.

Parameter 1: Upper bound to the number of added stiffness terms.

C.45 *DASHPOTS

Used to specify additional viscous damping terms at a node, to be added directly into the damping matrix.

Parameter 1: Upper bound to the number of added viscous damping terms.

C.46 *MASSES

Used to specify additional mass terms at a node, to be added directly into the mass matrix.

Parameter 1: Upper bound to the number of added mass terms.

C.47 *LINESEARCH

This parameter invokes the line search operation in the iterative process.

No parameter.

The maximum number of search distance iterations is internally set to five. When MHOST fails to detect the zero point, an appropriate message is printed and execution continues.

The current version of the MHOST code does not support this option when used in conjunction with the arc-length method. Any other solution option can be accelerated by invoking this option.

The operation involves multiple evaluation of the residual vectors and can be potentially time-consuming for some problems.

C.48 *SECANTNEWTON

When this option is flagged, secant Newton iteration is invoked.

No parameter.

The structure is almost identical to the BFGS update. The same potential economy is expected by utilizing this procedure.

This option cannot be used with the BFGS option. SECANTNEWTON is available for quasi-static and direct integration dynamic problems.

C.49 *HARMONICLOAD

This option is used to invoke harmonic nodal force loading for dynamic analysis. One integer parameter.

Parameter 1: Upper bound to the number of degrees-of-freedom at which harmonic loads are specified.

In addition, a continuation line must be provided with a single real parameter, specifying:

Parameter 1 (REAL): Frequency of excitation, in cycles/time.

C.50 *BASEEXCITATION

This option is used to invoke harmonic base excitation at one or more base locations. One integer parameter.

Parameter 1: Upper bound to the number of degrees-of-freedom at which harmonic base excitations are specified.

In addition, a continuation line must be provided with a single real parameter, specifying:

Parameter 1 (REAL): Frequency of excitation, in cycles/time.

C.51 *COMPOSITE

The composite laminate model is used for defining the characteristics of shell element type 75, when this parameter data is specified. This option overwrites the element parameter data (See Table ELIB.1) and, thus, must appear after the element type definition (*ELEMENTS) parameter line. No parameter.

Note that this option requires the nodal material data definition in the model data section using the LAMINATE option.

This option was developed from the formulation of M. D. Minich and C. C. Chamis (1975), Analytical Displacements and Vibrations of Cantilevered Unsymmetric Fiber Composite Laminates, NASA Technical Memorandum NASA TMX-71699. The underlying assumption is that the material exhibits linear elastic response. Any requests for inelastic constitutive laws are ignored in the code when this option is invoked. Creep and thermal strain options are available in conjunction with this option.

C.52 *STIFFENING

When this option is requested, stress stiffening due to initial stresses is invoked.

Parameter: Increment number at which stiffening is to be included. All increments following this increment will also have stiffening. (Defaults to increment number one).

C.53 *CENTRIFUGALMASS

When this option is requested centrifugal mass stiffness effects due to angular velocity are included. (Default to increment number one).

Parameter 1: Increment number at which centrifugal mass stiffness effects are to be included. All increments following this increment will also have centrifugal mass effects included.

The angular velocity is defined with the DISTRIBUTEDLOAD model data input. The axis about which the angular velocity occurs is defined with the BODYFORCE model data input.

C.54 *CONJUGATEGRADIENT

This parameter invokes the conjugate gradient method for the mixed finite element equations. The stiffness equations for the displacement method are used as the preconditioner. The line search is automatically turned on with this option. Not usable with thequasi-Newton and secant-Newton options. The algorithmic detail is found in the paper by Nakazawa, Nagtegaal and Zienkiewicz (1985), "Iterative Methods for Mixed Finite Element Equations", ASME AMD Vol. 73.

C.55 *NOECHO

This option suppresses the direct echo print of the bulk data input. This option reduces the amount of line printer output for large finite element models.

C.56 *POWERSPECTRUM

Used to invoke the driver for frequency-domain analysis of random vibration using power spectrum techniques. Three integer parameters may be specified:

Parameter 1: The number of discrete frequencies at which power spectral density values are specified.

Parameter 2: The number of breakpoints used to define the frequency ranges to be used in the frequency-domain integration.

Parameter 3: If a nonzero integer is specified, the spatial loading correlation function S_{XX} is assumed to be frequency-dependent, and is recomputed at each integration point in the frequency. Defaults to 0, i.e., S_{XX} is computed only once.

Both parameters are upper bounds to the actual numbers used in the analysis and will be adjusted if fewer entries are specified in the model data section. For further elaboration on the input required for power spectrum analysis, please refer to the POWER, SPECTRUM and BREAKPOINTS options in the model data section. See also user subroutine USXX for user-defined S_{XX}.

C.57 *PULSELOAD

This option may be used to define conveniently a force time-history which may be triggered repeatedly at a number of points at specified times. This capability may be used to model traveling pressure wave phenomena. Two integer parameters are required:

Parameter 1: The number of discrete points in time at which the pulse load time-history is specified.

Parameter 2: The number of occurrences of the pulse load at different times and locations in the course of the analysis.

Both parameters are upper bounds to the actual numbers used in the analysis and will be adjusted if fewer entries are specified in the model data section.

C.58 *PRESSURE

This option is used to invoke the nodal surface pressure distribution option for meshes assembled for 2D or 3D continuum-type elements. These include element types 3, 7, 10, 11 and 151 through 154.

No parameters are needed.

The algorithm is based on the assembly of nodal vectors corresponding to a unit pressure on all faces of the element. On a continuum mesh, these vectors cancel out at all internal nodes, producing non-zero vectors only at the external boundary nodes. The assembled vectors provide well-defined boundary normals, which can then be used for applying nodal pressures.

C.59 *HARDENING_SLOPE

This option specifies the number of data entries to define a piecewise-linear hardening slope.

Parameter 1: The maximum number of data entries in *WORKHARD data segment in the Model Data block.

Note that *WORKHARD and *YIELD are options available in the Model

Data block. *WORKHARD specifies elastic plastic constitutive behavior for the entire domain. *YIELD allows users to define the initiation of nonlinear material response at each node.

C.60 *LARGE_DISPLACEMENT

This parameter card invokes the large displacement option. Note that the updated Lagrangian algorithm is implemented to follow the geometry change.

Parameter 1: Increment number at which the large displacement terms are turned on.

C.61 *FINITE_STRAIN

This option invokes the finite strain terms in elastic-plastic computations. This parameter card cannot be specified without also specifying *LARGE_DISPLACEMENT card.

Parameter 1: Increment number at which the finite strain terms are turned on.

C.62 *FOLLOWER_FORCE

This option allows the loading condition to be updated following large displacement of the structure. This option cannot be used without also specifying the *LARGE_DISPLACEMENT card.

Parameter 1: Increment number at which this option is turned on.

C.63 *END

This indicates the end of the parameter data input.

D. MODEL DATA

The model data follows immediately after the *END line of the parameter data. It consists of a number of segments. Each segment starts with a keyword line with at most one integer on the line.

The number of data records following the keyword line does not have to be determined beforehand. The program reads data records until a new keyword line is encountered.

The current version of MHOST supports the following options:

SECTION D INDEX

KEYWORD	SECTION NO.	PAGE
*BASEXCITATION	D.32	49
*BEAMSECTION	D.26	46
*BODYFORCE	D.21	44-45
*BOUNDARY	D.2	36
*BREAKPOINTS	D.39	52-53
*COORDINATES	D.8	40
*DAMPING	D.24	45-46
*DASHPOTS	D.29	47-48
*DEBUG	D.15	43
*DISTRIBUTEDLOAD	D.19	43-44
*DMATRIX	D.34	50
*DUPLICATENODE	D.22	45
*ELEMENTS	D.1	36
*END	D.16	43
*ENDPOWER	D.40	53
*ENDPULSELOAD	D.44	54
*FOLLOWER_FORCE	D.50	55
*FORCES	D.4	37-38
*HARMONICLOAD	D.31	48
*HOLE	D.36	52
*INCLUDE	D.45	54
*INCREMENTS	D.10	40-41
*ITERATIONS	D.6	38-39
*LAMINATES	D.33	49-50
*MASSES	D.30	48
*NODALDEFINITION	D.43	53-54
*ORIENTATION	D.35	50-51
*POWER	D.37	52
*PRESSURE	D.49	54-55
*PRINTOPTION	D.13	41-42
*PROPERTIES	D.7	39-40
*PULSELOAD	D.41	53
*RESUME	D.46	54

*SAVE	D.14	43
*SHIFT	D.27	46-47
*SPECTRUM	D.38	52
*SPRINGS	D.28	47
*STOP	D.17	43
*STRESS	D.12	41
*TEMPERATURES	D.11	41
*TIME	D.23	45
*TIMEHISTORY	D.42	53
*TRANSFORMATIONS	D.5	38
*TYING	D.3	36-37
*UFXORD	D.20	44
*WORKHARD	D.18	42

D.1 *ELEMENTS

This data segment defines the element connectivity. When *EMBED option is flagged, the subelement mesh pattern can be defined by an additional integer entry in the data line. The number of data lines must be less than or preferably equal to the upper bound value given in the parameter data block.

One Parameter. Containing the element type number.

Subsequent data records each contain N+1 integers, with N being the number of nodes per element. If the element is being further refined into a subelement mesh, an (N+2)nd integer entry is also required.

Integer 1: Element number.

Integers 2 to N+1: Node numbers of the element.
Integer N+2 : Key for the subelement refinement.

If the element type has more than 15 nodes continuation lines must be used (see B.4).

D.2 *BOUNDARY

This data segment is used to impose the constraints on nodal displacements. The number of data lines must be less than or equal to the upper bound value given in the parameter data block.

No Parameters. Subsequent data records each contain two (2) integers and a real.

Integer 1: Node number.

Integer 2: Degree-of-freedom.

Real 1: Prescribed displacement value. (Default value is zero).

If multiple specifications occur for the same degree-of-freedom at the same nodal point, the last specified value is used.

D.3 *TYING

This keyword is used to set up the data segment for constraints between nodal displacement degrees-of-freedom. The number of constraint equations must be less than or equal to the upper bound value given in the parameter data block.

Each tying constraint is specified by two data records. A typical form of tying constraint is:

$$U_s = \sum_m A_m U_m$$

where s denotes the 'slave' degree-of-freedom ('tied' degree-of-freedom) and m denotes the 'master' degree-of-freedom ('retained' degree-of freedom). The first record specifies the nodes and degrees-of-freedom involved in the tying, the second record specifies the multiplication factors.

First Record:

Integer 1: Number of degrees-of-freedom involved in the tying.

Integer 2: Node number of the tied degree-of-freedom.

Integer 3: Degree-of-freedom number of the tied degree-of-freedom.

Integer 4: Node number of the first retained degree-of-freedom.

Integer 5: Degree-of-freedom number of the first retained degree-of-freedom.

Integer 6 to 2N+1:
Node numbers/degree-of-freedom numbers of subsequent retained degrees-of-freedom (if N > 2).

NOTE: Not more than 16 integers should be entered on each data line (see B.4).

Second Record:

Real 1: Multiplication factor of the first retained degree-of-freedom in the tying equation.

Real 2 to N-1:
Multiplication factor for a subsequent retained degree-of-freedom in the tying equation (if N>2).

Real N: Prescribed value for a non-homogeneous tying equation.

NOTE: Not more than eight (8) reals should be entered on one data line (see B.4).

If the same degree-of-freedom appears as the tied degree-of-freedom in more than one tying equation, the last prescribed tying equation applies. Note that tying constraints take precedence over boundary conditions (nodal displacement constraints).

D.4 *FORCES

This keyword is used for the data segment of nodal force vectors. The number of data lines must be less than or equal to the upper bound value specified in the parameter data block.

No Parameters. Subsequent data records each contain two (2) integers and one (1) real.

Integer 1: Node number.

Integer 2: Degree-of-freedom.

Real 1 : Magnitude of the applied point load.

If multiple specifications occur for the same node and degree-of-freedom the last specified magnitude is used.

D.5 *TRANSFORMATIONS

This data segment specifies the coordinate transformation for the nodal degree-of-freedom. Only the rotation can be specified by this option.

No Parameters. Subsequent data records each contain two (2) integers and one (1) real

Integer 1: Number of the node on whose coordnates the transformation is applied.

Integer 2: Axis number around which the coordinate system is rotated (if the node is two-dimensional, enter a zero).

Real 1: Angle over which the coordinate system is rotated (in degrees).

If more than one coordinate transformation is applied at a node, the program executes all the entered transformations successively. This feature can be used to obtain transformations around an arbitrary axis. It is also possible to use tying in order to apply boundary conditions in an arbitrary direction.

Each subsequent transformation acts upon the last previous coordinate system defined at the node, not the original coordinate system used for the data input.

Note that this option follows the right hand coordinate convention.

D.6 *ITERATIONS

This data segment control the iterative solution process. When the BFGS option is used, the maximum number of iteration must be less than or equal to the number of BFGS vectors (default is 10). When the subelement iteration is invoked, convergence of the subelement solution is tested with respect to the normalized relative displacement.

No Parameters. One data line, containing one (1) integer and four (4) reals.

Integer 1: Maximum number of iterations allowed.

Real 1: Maximum allowable relative error in the residuals.

Real 2: Maximum allowable absolute error in the residuals.

Real 3: Maximum allowable relative error in the root mean square of displacement (L matrix norm of displacement vector).

Real 4: Maximum allowable relative error in the root mean square of strain energy associated with residual (energy semi-norm of the solution).

The default for the maximum number of iterations is 5. The default for the relative allowable error is 5.0. All other convergence criteria are inactive unless the tolerance is specified by the user with a non-zero positive number.

Iteration continues until all active convergence criteria are satisfied and the next incremental solution is entered unless the last increment is reached.

When the solution fails to converge after the maximum number of iterations is reached, the code prints out the solution for the last iteration and then terminates the execution with ERROR logo on the output files.

D.7 *PROPERTIES

This data segment is used to specify the linear elastic material property data at nodes.

One (1) parameter, containing the element type.

Subsequent data records contain two (2) integers and N reals, where N is the number of properties specified for this element type in Section G.

Integer 1: First node in a consecutive series of nodes (dummy).

Integer 2: Last node in this consecutive series (dummy).

Reals 1 to N:
Properties of the series of nodes.

NOTE: If N > 7, continuation lines must be used. (See B.4).

The element property data must be read after the element type is defined in the same input data deck.

D.8 *COORDINATES

This data segment specifies the coordinates of nodal points in the mesh. The total number of entries must be less than or, preferably equal to, the maximum number specified in the parameter data block.

The coordinate system in which the location of nodal points is defined is a global one not subjected to any transformation.

One (1) parameter, containing the number of coordinates to be read in for this node. If not specified, the maximum number of coordinates for all element types as specified under C.1 is used.

Subsequent data records contain one (1) integer and N reals, where N is the number of coordinate directions specified or the default.

Integer 1: Node number.

Reals 1 to N:
Coordinates of the node.

NOTE: if N > 7, continuation lines must be used (see B.4).

For the shell element (Element Type 75), four real entries are required in each data line. The first three real entries are the global coordinates of the current nodal point with the fourth entry being the thickness of the shell. If the thickness is left unspecified, the value in the previous data line is automatically substituted. At least the first data line must contain the shell thickness entry.

For the two-node beam element (Element Type 98), six real data entries are required in each data line. The first three real data entries are the three global node coordinates. The second three real data entries are the three global components of the unit vector defining the direction of the beam cross-section y-axis.

D.10 *INCREMENTS

This data segment is used to specify the maximum number of increments. The actual number of increments are usually less than the value specified here. The incremental process continues until the maximum number is met, *STOP is encountered or no other increment control parameter is specified. Usually, it is not necessary to include this data segment in an input data file.

No parameters. Followed by one data line containing one (1) integer.

Integer 1: Maximum number of increments.

The default is 30.

D.11 *TEMPERATURES

The nodal point temperatures for the initial condition (zeroth increment) are specified by this data segment. The reference temperature is zero (0) unless otherwise specified by a user subroutine. An error message is issued when this model data option is input without the TEMPERATURE parameter card.

For the shell element (Element Type 75), the temperature has to be given at every layer for pre-integration through the thickness.

No parameter. Subsequent data records contain two integers and N reals, where N is the number of temperatures to be specified at nodes. N is one for all the elements except for the shell element where N is the number of layer integration points through the thickness of the shell.

Integer 1: The first node number

Integer 2: The last node number

Real 1 to N: Temperature at the nodes

D.12 *STRESS

The MHOST program allows specification of stress values at nodes. This operation is inconsistent with the variational formulation and, hence, it is not recommended under normal situations. The imposition of the stress boundary condition may lead to unstable iteration in the mixed process.

With this option, stress components can be prescribed at nodal points (for the mixed iteration only). The computed nodal values are replaced by the prescribed values. No parameters. Subsequent data records each contain two (2) integers and a real.

Integer 1: Node numbers

Integer 2: Stress component number

Real 1: Prescribed value

D.13 *PRINTOPTION

This option specifies the generation of lineprinter output. The contents of the lineprinter output is determined by subsequent data records. Once the print output is defined, the same output can be obtained in subsequent increments by entering the keyword line only.

Upon reading of the *PRINTOPTION data, the state currently existing is printed at the end of the increment. A new specification can be entered at any stage: the old print specification is then completely overwritten.

The print specification consists of keywords and sequences. Two keyword groups can be distinguished; keywords describing the quantities to be printed:

STRESS
STRAIN
PLASTIC
THERMAL
CREEP
INCREMENTALDISPLACEMENT
TOTALDISPLACEMENT
FORCE
REACTION/RESIDUAL
EQUIVALENT
TEMPERATURE

and keywords describing the finite element model

ELEMENT
INTEGRATIONPOINT
NODE
LAYER

Only the first four characters of a keyword need to be specified. A keyword of the first group can be followed by one to three keywords of the second group. A particular keyword of the second group may occur only once, however. The combinations must be appropriate. For instance, an element range should not be specified for the printing of displacements.

Each keyword may be followed by 1 or 2 numbers, which specify a sequence. For keywords in the first group, this sequence defines the components to be printed. If no sequence i specified, all components are printed. If a keyword of the first group is not followed by any keywords of the second group the default printout of nodal values is generated.

The number of keywords in the first group (printsets) may not be larger than specified in the *PRINTSET parameter segment, or, if absent, larger than the default of 10.

D.14 *SAVE

This option controls the generation of the restart file at the end of increment zero.

No parameters are specified.

If the run is a restart run, the new save file will overwrite the old restart file.

D.15 *DEBUG

This option controls the debug print. When this keyword line is present, all element stiffness matrices are printed in the line printer output file.

The amount of information produced by this option could be extremely large. Recommended to use only for small meshes.

D.16 *END

This indicates the end of the model data input.

D.17 *STOP

With this option, program execution is terminated immediately. If it is the user's intention only to check the input data, the option can be included before any chosen *END line. It can also be used for termination of plot or print specification. In that case, the analysis is terminated immediately after generation of a plot or print file.

D.18 *WORKHARD

This segment must be entered when an elastic-plastic material is specified by CONSTITUTIVE in the parameter data block.

One parameter, which specifies the number of data points used to specify the stress-strain law. The stress-strain relation is given as equivalent stress versus equivalent plastic strain, one pair of data points per card. Note that the first pair will specify the yield stress, at zero plastic strain. A maximum of 50 data lines can be read in by this option. Data entered in triples as:

REAL 1: equivalent stress
REAL 2: equivalent plastic strain
REAL 3: equivalent shift for kinematic and combined hardening

D.19 *DISTRIBUTEDLOAD

This option defines gravity, centrifugal and traction-type distributed loadings. The keyword has one parameter input, the element type. This is followed by at least one data line to specify the loading condition. The format is:

Integer 1:	the first element of a series of elements subjected to traction loading.
Integer 2:	the last element of the series.
Integer 3:	the index for the loading type.
REAL 1:	intensity of loading type 1 for the current element.
REAL 2:	intensity of loading type 2 for the current element.
REAL 3:	intensity of loading type 3 for the current element.

See Section G, ELEMENT LIBRARY, for details of the distributed load input for a given element type.

D.20 *UFXORD

Two integer parameters, which specify the range of node numbers for which user subroutine UFXORD is used to generate coordinates.

The user subroutine must be in the load module when MHOST is executed with this option.

D.21 *BODYFORCE

To invoke body force loading, including both gravity and centrifugal loadings. One integer parameter IBODY to select an option:

1. the gravity force
2. the centrifugal force

with one or two data containing two or three real parameters.

If the parameter IBODY=1, one data line of real numbers to specify a direction vector for the gravity acceleration. The data line is:

REAL 1:	x component of gravity direction vector.
REAL 2:	y component of gravity direction vector.
REAL 3:	z component of gravity direction vector.

If the parameter IBODY=2, two data lines of real numbers are needed to specify a pair of points to define the axis of rotation. Each line consists of a set of the coordinate data. The data line is:

REAL 1: x coordinate of a point on the axis.

REAL 2: y coordinate of a point on the axis.
REAL 3: z coordinate of a point on the axis.

D.22 *DUPLICATENODE

This parameter flags the duplicate node option input. Subsequent data records each contain 2 integers.

Integer 1: Slave node number

Integer 2: Master node number

D.23 *TIME

Enter time increment and total time for either creep or transient analysis, to be followed by one data line containing two integers and two reals.

Integer 1: Maximum number of time increments.

Integer 2: Flag for adaptive time step selection (non-zero value is required to invoke this option).

REAL 1: Size of time increment.

REAL 2: Total period of time for the analysis.

D.24 *DAMPING

With this option structural damping is introduced for transient dynamic calculations. One data line containing two real parameters is required.

REAL 1: Coefficient for the mass matrix (w , see below)

REAL 2: Coefficient for the stiffness matrix (w , see below)

Damping is introduced to the equation of motion

$$M\,a + C\,v + K\,u = F$$

where

$$C = w_1\,M + w_2\,K$$

Note that K is updated whenever the material tangent is recalculated and, therefore, C will also be updated.

If the mode superposition option for linear dynamics analysis has been flagged by specifying a number in the continuation line to the

*MODAL option in the parameter data section, modal damping is assumed. In this case, instead of the format specified above, one or more data lines each containing two integers and one real are expected. Data lines will specify:

Integer 1: First mode in a consecutive series.

Integer 2: Last mode in the same consecutive series. (Defaults to the first mode specified).

Real 1: Damping ratio for modes in the series. The resulting uncoupled equations of motion will be

$$q_r + 2b_r w_r q_r + w_r^2 q_r = \frac{1}{m_r} \{D_r\}^T \{f\}$$

where b_r is the damping ratio for the r-th mode.

D.26 *BEAMSECTION

This option is used to specify one or more sets of cross-sectional properties for the elastic beam elements. Note that beam cross-sections are defined on a nodal basis and not on an element basis. Subsequent data records contain two integer and six real values, specifying:

Integer 1: First node in a consecutive series.

Integer 2: Last node in the same consecutive series.

Real 1: Shear Area in the local x-direction.

Real 2: Shear Area in the local y-direction.

Real 3: Cross-Sectional Area.

Real 4: Moment of Inertia about the local x-axis.

Real 5: Moment of Inertia about the local y-axis.

Real 6: Torsional Constant.

For additional information on the definition of section properties, refer to the beam element description in Section G.

D.27 *SHIFT

Used to control the power shifts during eigenvalue extraction for linear dynamic analyses. At least one data line must follow, specifying:

Integer 1: Maximum number of eigenvalues to be extracted within the given frequency range.

Real 1: Lower bound for the frequency range, in cycles/time.

Real 2: Upper bound for the frequency range, in cycles/time.

NOTE: The sum of the maxima specified in the first integer value for all power shifts must be less than or equal to the number of modes specified in the MODAL card.

Any eigenvalues found outside the prescribed frequency ranges will be discarded and the program will check whether all the eigenvalues within each range have been found. A warning message is printed whenever there is the possibility of more than the specified maximum number of eigenvalues being present within a given frequency range. If more than one frequency range is prescribed, MHOST stores the new eigenvalues and eigenvectors immediately after all previously searched frequency ranges.

NOTE: Specification of two overlapping frequency ranges will result in all modes present within the overlap zone being extracted and stored twice.

D.28 *SPRINGS

Specifies additional stiffness values at a node that are to be added directly into the stiffness matrix. This option may be used, for example, to introduce springs to the ground, or a foundation stiffness. Subsequent data lines contain:

Integer 1: Node number for the added stiffness.

Integer 2: Degree-of-freedom at the node.

Integer 3: Second degree-of-freedom at the node, used to specify off-diagonal terms. Defaults to the first degree-of-freedom specified (uncoupled springs).

Real 1: Magnitude of the stiffness term to be added at this degree-of-freedom.

NOTE: If a second degree-of-freedom is specified, it must be greater than or equal to the first one. That is to say, only the upper-triangular off-diagonal terms can be entered.

D.29 *DASHPOTS

Specifies additional viscous damping terms at a node, to be added directly into the damping matrix. Subsequent data lines contain:

Integer 1: Node number for the added damping.

Integer 2: Degree-of-freedom at the node.

Integer 3: Second degree-of-freedom at the node, used to specify off-diagonal terms. Defaults to the first degree-of-freedom specified (uncoupled dampers).

Real 1: Magnitude of the viscous damping term to be added at this location.

D.30 *MASSES

Specifies additional mass terms at a node, to be added directly into the mass matrix. This option may be used to include lumped masses not included in the finite element discretization. Subsequent data lines contain:

Integer 1: Node number for the added mass.

Integer 2: Degree-of-freedom at the node.

Integer 3: Second degree-of-freedom at the node, used to specify off-diagonal terms. Defaults to the first degree-of-freedom specified (no coupling of mass).

Real 1: Magnitude of the mass term to be added at this location.

NOTE: Same as for *SPRINGS options, above.

D.31 *HARMONICLOAD

This option is used to prescribe harmonic nodal force loading in the form of

$$f = \bar{f} e^{i(\omega t + \Delta)}$$

where ω is the single excitation frequency entered in the continuation line to the HARMONICLOAD option in the parameter data block. Subsequent data records contain two integers and two reals, specifying:

Integer 1: Node number for the applied load.

Integer 2: Degree-of-freedom at the node.

Real 1: Magnitude f of the applied load.

Real 2: Phase angle D of the applied load. The phase angle value defaults to zero, which corresponds to a cos(ω)t loading.

D.32 *BASEEXCITATION

This option is used to prescribe harmonic base excitation in the form of

$$u = \bar{u}e^{i(\omega t + \Delta)}$$

at one or more base locations, where w is the single excitation frequency entered in the continuation line to the *BASEEXCITATION option in the parameter data section. Subsequent data records contain two integers and two reals, specifying:

Integer 1: Node number for the imposed base excitation.

Real 1: Amplitude u of the excitation.

Real 2: Phase angle D of the excitation. The phase angle value defaults to zero, which corresponds to cos wt excitation.

NOTE: A base excitation can only be applied at an unconstrained degree-of-freedom. If the model is to be excited simultaneously at all base points, it must be modeled as a free body during eigenvalue extraction.

D.33 *LAMINATES

This option defines material properties necessary to analyze composite laminates modeled with the MHOST shell element. The COMPOSITE option in the parameter data block must be input to properly allocate memory. No parameter.

The internal consistency of the input data stream for this option is not tested in the current version of MHOST. Inconsistent specification of data may result in an execution error or unreliable results.

The material description here consists of four data lines with formats as follows:

Line 1 Integer 1: The first nodal point identification of the series of nodes at which the material modulus is given by the following three data lines.

Integer 2: The last node of the series.

Line 2 Reals 1 to 8: Entries of material modulus for membrane terms.

Line 3 Reals 1 to 8: Entries of material modulus for membrane-bending coupled terms.

Line 4 Reals 1 to 8: Entries of material modulus for bending terms.

The above numeric data set must be repeated for all nodal points.

The entries are:

Real 1 - local s_{xx} - e_{xx} component of the material moduli.

2 - local s_{xx} - e_{yy} component of the material moduli.

3 - local s_{xx} - e_{xy} component of the material moduli.

4 - local s_{yy} - e_{yy} component of the material moduli.

5 - local s_{yy} - e_{xy} component of the material moduli.

6 - local s_{xy} - e_{xy} component of the material moduli.

7 - local s_{xz} - e_{xz} component of the material moduli.

8 - local s_{yz} - e_{yz} component of the material moduli.

where local indicates the nodal coordinate system defined at each individual nodal point.

D.34 *DMATRIX

Entries to the material matrix can be read in directly by invoking this option for continuum elements (element types 3, 7, 10, 11, 101, 102 and 103). No parameter.

The material description records consist of NELSTR + 1 data lines. (NELSTR = number of element stress components). The format is:

Line 1 Two integer data:

Integer 1: The first nodal point identification number of nodes at which the entries of DMATRIX are given by the following NELSTR data lines.

Integer 2: The last node of the series.

Lines 2 to NELSTR NELSTR real data on each line.

Reals 1 to NELSTR: The entries of the current row of the DMATRIX.

D.35 *ORIENTATION

This option is used to define the preferred orientation of the material axes for anisotropic materials. Subsequent data lines contain two integers and three reals:

Integer 1: First node of a series of consecutive nodes.

Integer 2: Last node of a series of consecutive nodes.

Real 1: Rotation angle , in radians

Real 2: Rotation angle , in radians

Real 3: Rotation angle , in radians

The material axes for all nodes in the series will be oriented by a composition of three orthogonal transformations: (1) a rotation of about the Z-axis, followed by (2) a rotation of f about the new y-axis, and by (3) a rotation of q about the final x-axis. These transformations are illustrated in Figure D.1. The righthand rule for rotations is used throughout.

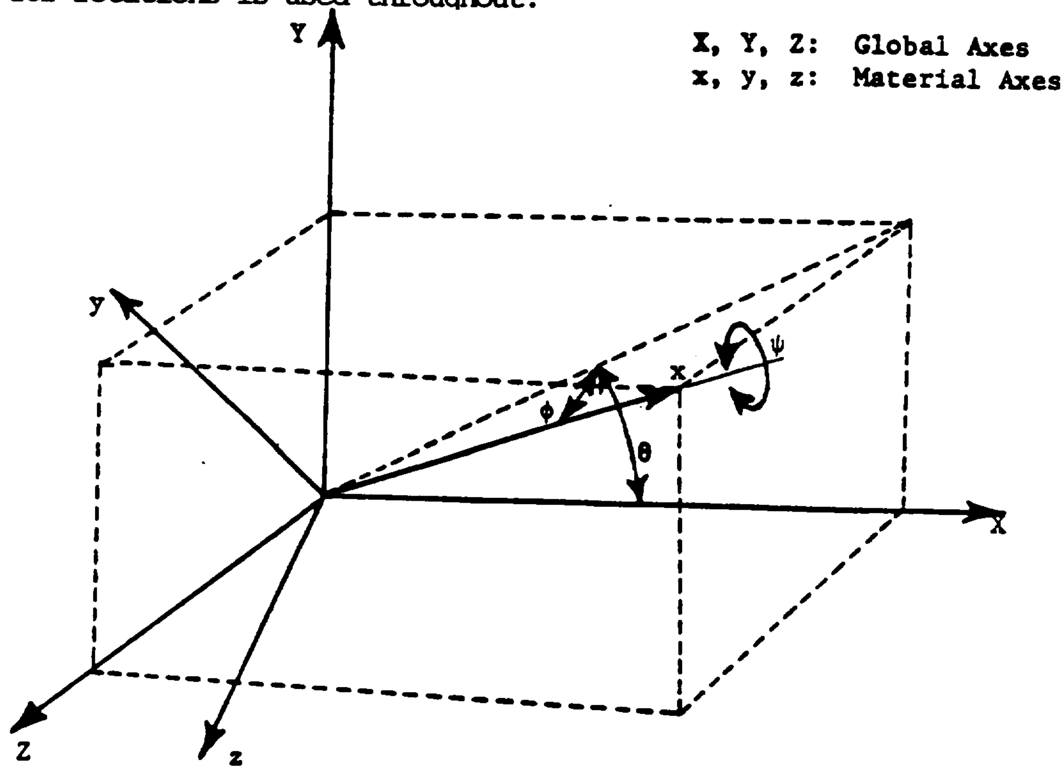

FIGURE D.1 Coordinate transformations for anisotropic material axes orientation

D.36 *HOLE

This keyword line defines an embedded hole in a global element. The *EMBED line must be included in the parameter data section. No parameter.

The following data lines consist of two integer and one real variables.

Integer 1: The identification number of the element in which a hole is embedded.

Integer 2: The subelement type. If an element type other than the global element type is specified here, the additional subelement type must be included in the Parameter Input Block.

Real 1: The dimensionless diameter of the embedded hole. The value is the diameter of the hole in the isoparametric space associated with the global element.

The number of subdivisions in the subelement mesh for the embedded hole is explicitly coded in the subroutine HOLEIN. The recommended subelement division of two meshes along each element edge and two meshes in the radial direction is provided with the installation. This option is limited to two-dimensional analysis only.

D.37 *POWER

This option is used to specify the input for frequency-domain analysis using the power spectrum technique. It must be followed by the SPECTRUM and BREAKPOINTS input options and terminated with the ENDPOWER option. No parameters are required.

D.38 *SPECTRUM

This option is used to define the power spectral density function (PSD) to be used in the analysis. Subsequent data records contain two real values:

Real 1: The frequency (in radians/time).

Real 2: The PSD value at the frequency above.

The frequencies must be input in ascending order, since linear interpolation will be used to evaluate PSD values at intermediate points.

D.39 *BREAKPOINTS

This option is used to specify the breakpoints defining the frequency ranges over which the frequency-domain integration is performed. Subsequent data lines contain a single real value:

Real 1: Frequency breakpoint (in radians/time).

The spectral density function for modal response will be integrated over the frequency range between each pair of successive breakpoints using 5-point Gaussian quadrature. For best results, these breakpoints should be specified close together, in the neighborhood of each undamped natural frequency of the system.

By specifying a pair of breakpoints at $\omega_n - \delta$ and $\omega_n + \delta$ where ω_n is a resonant frequency and d is a small deviation, the response "peak" for the ω_n frequency should be integrated accurately. The frequency breakpoints must be specified in ascending order.

NOTE: The spatial correlation function S_{XX} between the Fourier spectra of the random loading at two points must be specified via the user subroutine USXX.

D.40 *ENDPOWER

This option terminates the input data block for power spectrum data and returns control to the model data reader.

D.41 *PULSELOAD

This option is used to specify the input for the pulse load definition. It must be followed by the TIMEHISTORY and NODALDEFINITION options and terminated with the ENDPULSELOAD option. No parameters are required.

D.42 *TIMEHISTORY

This option is used to define load time-histories. Subsequent data lines contain two real values:

Real 1: Time since the start of the pulse load

Real 2: The value of the load at the time above.

The times must be input in ascending order since linear interpolation will be used to evaluate the value of the loading at intermediate times. For optimal results, the pulse load time-history should be defined at times which are integer multiples of the time-step size used for the analysis.

D.43 *NODALDEFINITION

This option is used to define the locations and times at which the pulse load time history is to be triggered in the course of the analysis. Subsequent data lines contain two integers and two reals:

Integer 1: Node number at which the pulse load will occur.

Integer 2: Degree-of-freedom in which the pulse load will act.

Real 1: The time at which the pulse load time history will be triggered at the node.

Real 2: A scale factor for the time history at the node. It defaults to the last scale factor defined. If the first line has no scale factor defined, it will default to a value of 1.0 (no scaling).

D.44 *ENDPULSELOAD

This option terminates the input data block for pulse load data and returns control to the model data reader.

D.45 *INCLUDE

This option is available in the versions of MHOST compiled under FORTRAN-77 and allows temporary input redirection to a second input file. It must be followed by a line containing the file name of the secondary input file. Upon reaching this option, MHOST will open the secondary input file and continue reading input from it until a *RESUME card or end-of-file is found. This option should NEVER be used within the secondary input file, but it may be used more than once in the main input file, provided that *RESUME cards are used to terminate all secondary input files.

D.46 *RESUME

This option is available in the versions of MHOST compiled under FORTRAN-77 and allows redirection of input back into the main input file (FORTRAN unit 5). This option should only be used within secondary input files accessed via the *INCLUDE option.

D.49 *PRESSURE

On meshes assembled from continuum-type elements, this option may be used to specify nodally defined pressures over a range of external boundary nodes. This option is not available for meshes comprising beam or shell-type elements.

No parameters are needed.

Subsequent data lines contain two (2) integers and one (1) real:

Integer 1: The first of a series of nodes to which nodal surface pressures are applied.

Integer 2: The last node in the series.

Real 1: Intensity of pressure loading, defined as positive into the body. Nodal pressures are specified in units of force per unit area on 3D and axisymmetric problems, and force per unit length for plane strain and plane stress problems.

D.50 *FOLLOWER_FORCE

This keyword is used for the data segment of nodal follower force vectors in conjunction with the large displacement option.

The number of data lines must be less than or equal to the upper bound value specified in the parameter data block.

No Parameters. Subsequent data records each contain two (2) integers and one (1) real.

Integer 1: Node number.

Integer 2: Degree-of-freedom.

Real 1: Magnitude of the applied point load.

If multiple specifications occur for the same node and degree-of-freedom the last specified magnitude is used.

E. INCREMENTAL DATA INPUT

The general comments made in Section D regarding structure of the model data input also apply to the INCREMENTAL DATA input. All quantities entered in this data block, except the temperature data, are INCREMENTAL quantities, which are used to update existing values. If certain incremental quantities in this block are not specified, the program increments the quantity with the same value as in the preceding increment.

The following value specified in the options are available to define incremental data:

SECTION E INDEX

KEYWORD	SECTION NO.	PAGE
*ACCELERATION	E.20	60
*AUTOINCREMENT	E.9	58
*BODYFORCE	E.16	59
*BOUNDARY	E.1	57
*DEBUG	E.11	58
*DISPLACEMENT	E.18	59
*DISTRIBUTEDLOAD	E.4	57
*END	E.15	59
*ENDINITIALCONDITION	E.22	60
*FOLLOWER_FORCE	E.23	61
*FORCES	E.3	57
*INITIALCONDITION	E.17	59
*ITERATIONS	E.7	57-58
*PERIODICLOADING	E.21	60
*PRINTOPTION	E.12	58-59
*PROPORTIONAL	E.8	58
*SAVE	E.10	58
*STOP	E.14	59
*STRESS	E.6	57
*TEMPERATURES	E.5	57
*TIME	E.13	59
*TYING	E.2	57
*VELOCITY	E.19	59-60

E.1 *BOUNDARY

Input is the same as in the model data block. If the specified boundary condition existed in a preceding increment(s), the boundary condition is updated. If no boundary condition was specified in a preceding increment(s), a new boundary condition is added. Boundary conditions can be removed by specifying the appropriate node number and degree-of-freedom number, with the node number preceded by a minus (-) sign. The boundary condition is then removed from the list. Removing boundary conditions creates space for additional boundary conditions, up to the maximum limit defined in the parameter data.

E.2 *TYING

Input is the same as in the model data block. If the tied degree-of-freedom was specified in a preceding increment(s), the tying for the degree-of-freedom is replaced by the newly specified tying. If the degree-of-freedom was not specified in the preceding increment(s), a new TYING equation is added.

Existing tying can be removed by specifying a tying equation with the relevant degree-of-freedom included as the tied degree-of-freedom but, without any retained degrees-of-freedom. This can be achieved by setting the number of degrees-of-freedom in the tying input equal to one. If a tying is removed, no data record specifying coefficients is required.

E.3 *FORCES

Input is the same as in the model data block. See additional comments under E.1 on updating the data.

E.4 *DISTRIBUTEDLOAD

Input is the same as in the model data block. See additional comments under E.1 on updating the data.

E.5 *TEMPERATURES

Input is the same as in the model data block. Previously specified temperatures are overwritten.

E.6 *STRESS

Input is the same as in the model data block. See additional comments under E.1 on updating the data.

E.7 *ITERATIONS

Input is the same as in the model data block. This option makes it possible to modify the iteration parameters.

E.8 *PROPORTIONAL

No parameters. This option allows multiplication of all correctly applied load increments (pressures, forces, prescribed displacement and non-homogenous tying equations) with a given factor. Thermal loads and body forces are not affected by this option. The multiplication only applies to loads which

1. were read in a previous incremental data block, and

2. load increments which were read within the current block PRIOR to the PROPORTIONAL option.

One data line containing one real.

Real 1: Multiplication factor.

E.9 *AUTOINCREMENT

This option invokes application of the automatic load increment procedure. Existing incremental loads are first modified according to the input entered within the same incremental data block. The adaptive load increment procedure, based on the arc length method, automatically corrects the load increment size at every iteration step.

One continuation line containing two integers.

Integer 1: Number of increments for which the specified load is applied. With adaptive load increments, the total load is equal to this number times the specified load.

Integer 2: Maximum number of adaptive increments. If not specified, the increment will not be adaptive.

E.10 *SAVE

Input is the same as in the model data block. This option writes the results computed by MHOST at the end of the current increment onto a restart file.

E.11 *DEBUG

Input is the same as in the model data block. With this option debug printout is generated.

E.12 *PRINTOPTION

Input is the same as in the model data block. With this option line printer output of the MHOST results is generated.

E.13 *TIME

Enter time increment and total time for either a creep or dynamic analysis. One data line containing two integers and two reals.

Integer 1: Maximum number of time increments.

Integer 2: Flag for adaptive step selection.

REAL 1: Size of time increment

REAL 2: Total time period to be covered.

E.14 *STOP

With this option the analysis is terminated immediately.

E.15 *END

This option indicates the end of the current incremental data set. If an incremental data block only contains the *END option, a proportional increment of one is taken. When using the AUTOINCREMENT option, one END card must be input for each of the increments.

E.16 *BODYFORCE

Input is the same as in the model data block. This option is used to define the body force load increment.

E.17 *INITIALCONDITION

This option is used to specify initial values for transient dynamic computations followed by options E.18 to E.21. This option must be concluded with ENDINITIALCONDITION. No parameters.

E.18 *DISPLACEMENT

This option is used to specify the initial nodal displacements for transient dynamic computations.

Subsequent data records contain one integer to identify the nodal point and N reals, with N being the number of displacement degrees-of-freedom.

E.19 *VELOCITY

This option is used to prescribe the initial nodal velocities for transient dynamic computations.

Subsequent data records contain one integer to identify the nodal point and N reals, with N being the number of displacement degrees-of-freedom.

E.20 *ACCELERATION

This option is used to prescribe the initial nodal accelerations for transient dynamic computations.

Subsequent data records contain one integer to identify the nodal point and N reals, with N being the number of displacement degrees-of-freedom.

E.21 *PERIODICLOADING

This option is used to prescribe periodic loading and displacement constraint in the form of

$$f = f \sin \omega t$$

for forces, and

$$u = u \sin \omega t$$

for displacements.

Subsequent data records contain three integers and two reals. These are:

Integer 1: Nodal point identification.

Integer 2: Flag for the nodal force/displacement specification.

1: nodal displacement.
2: nodal force.

Integer 3: Index for the constrained degree-of-freedom (the nodal force vector component).

Real 1: Period of cyclic loading.

Real 2: Intensity of cyclic loading.

E.22 *ENDINITIALCONDITION

This option terminates the initial condition input data block. It must be included if the INITIALCONDITION option is used.

E.23 *FOLLOWER_FORCE

Input is the same as in the model data block. See additional comments under E.1 on updating the data.

F. USER SUBROUTINES

User subroutines constitute one of the real strengths of the MHOST program. User subroutines allow the user to substitute his own subroutines for several existing routines in the program. A description of each of the available user subroutines is given in this manual. In addition, discussions of special routines are also included.

The IMPLICIT REAL*8 (A-H, O-Z) line is required in a user subroutine when the code is executed on 32 bits/word machines (e.g., VAX, IBM and PRIME). CRAY versions of MHOST code should not contain this declaration line.

It is recommended that a user not read data cards in most of the user subroutines. The reason for this is that the user subroutines will be placed into the nonlinear analysis loop, where the user cannot know how many times per increment the routine will be called.

The following user subroutines are operational in the Version 4.2 of the MHOST program:

SUBROUTINE NAME		PAGE
F.6	ANPLAS	70-71
F.7	CRPLAW	72-73
F.10	UBOUN	77
F.5	UCOEF	69
F.1	UDERIV	63
F.2	UFXORD	65
F.11	USXX	78
F.3	UTEMP	66
F.8	VSWELL	74
F.9	WALCON	75-76
F.4	WKSLP	67-68

F.1 SUBROUTINE UDERIV

In the MHOST program additional element types may be defined by the user. This is performed by giving integers 1 through 12 in C.1 and by coding subroutine UDERIV. This routine is called on an element level. The description of this routine is as follows:

```
      subroutine uderiv(beta,vjacob,coor,xrl,xirl,char,
     * iel,ic,jinc,jiter,nelcrd,nelnfr,nelnod,nelstr,nelint,nelchr,kw,
     * ierr,neltem,idf,jlaw,igaus )
c
c* * * * * *
c
c     element derivatives routine
c
c     beta    strain displacement matrix
c     vjacob  determinant
c     coor    nodal coordinates
c     xrl     total displacements of the nodes of an element
c     xirl    incremental displacements of the nodes of an element
c     char    element properties
c     iel     element number
c     ic      element type
c     jinc    increment number
c     jiter   iteration number
c     nelcrd  number of coordinate directions per node
c     nelnfr  number of degrees of freedom per node
c     nelnod  number of nodes per element
c     nelstr  number of stress/strain components per integration point
c     nelint  number of integration points per element
c     nelchr  number of properties per element
c     kw      output device
c     ierr    error flag
c     neltem  not used
c     idf     product of number of nodes per element and number of
c             degrees of freedom
c     jlaw    constitutive law type see c1.1
c     igaus   if reduced itegration set to 1 , otherwise equal to nelint
c
c* * * * * *
c
      implicit real*8 (a-h,o-z)
      dimension beta(nelstr,idf,nelint),vjacob(nelint),
     * coor(nelcrd,nelnod),xrl(nelnfr,nelnod),xirl(nelnfr,nelnod),
     * char(nelchr)
c* * * *
c
c     the user needs to define :
c
c     beta (strain displacement matrix) for all integration points
```

```
c     vjacob value of the jacobian at all integration points
c
c* * * *
      return
      end
```

F.2 SUBROUTINE UFXORD

In the MHOST program, the user may define a global coordinate system other than the rectangular Cartesian system by means of user subroutine UFXORD.

UFXORD may be used to modify or add to coordinate input through the COORDINATE option. UFXORD may also be used as an internal coordinate generator. The user must input the model definition card UFXORD, followed by a card giving the first and last numbers of a series of nodes for which UFXORD will be used. The program will then call UFXORD for each node in the series, so that the coordinates of that node may be modified or generated. The UFXORD option may be repeated as many times as necessary.

Subroutine UFXORD is written with the following header cards:

```
      subroutine ufxord(xord,ncrd,n)
c
      implicit real*8( a - h , o - z )
c
      dimension xord(ncrd)
c
c     xord  nodal coordinate to be defined or modified
c     ncrd  number of coordinate directions
c     n     node number
c
      return
      end
```

where:

NCRD	is the number of coordinates per node.
XORD	is the array of coordinates in the Nth node and will be passed in containing coordinates previously generated at the Nth node by COORDINATE, FXORD or UFXORD blocks.
N	is the node number.

One can use this routine to generate special coordinate systems (e.g., cylindrical or spherical) or to convert from special coordinate systems to a rectangular system.

F.3 SUBROUTINE UTEMP

In the MHOST program the user can define temperature dependent material properties by adding the user subroutine UTEMP as follows:

```
c ... subroutine utemp ... temperature-dependent material properties
c
      subroutine utemp(value,iflag,node,temp)
c
c ***********************************************************************
c
c     this user routine is used to specify material properties as
c     function of temperature for temperature-sensitive materials
c
c     input
c     -----
c     value    property value specified in the input deck
c     iflag    flag to identify the specific property
c              = 2  young's mudulus
c              = 3  poisson's ratio
c              = 4  coefficient of thermal expansion
c              = 6  shear modulus for cubic crystal
c     node     node number
c     temp     temperature
c
c     output
c     ------
c     value    property at the specified temperature as
c              overwritten by the user
c
c     this routine is called by subroutines 'hooklw' and 'thrstn'
c
c ***********************************************************************
c
      implicit real*8 (a-h,o-z)
c
c
      return
      end
```

An example of this user subroutine is found in example problem 1.4 of Volume II.

F.4 SUBROUTINE WKSLP

This subroutine makes it possible for the user to program the yield stress and the corresponding workhardening slope directly as a function of equivalent plastic strain and temperature. The user needs to define the value of the slope of the equivalent stress vs. equivalent plastic strain curve. The current yield stress may be defined also. The specification of the latter is optional. If the value of the current yield is not given, the program will calculate it from the initial yield value and the workhardening slopes defined in this routine.

Subroutine WKSLP is written with the following header cards:

```
      subroutine wkslp(ep,sy,h,shift,temp,intnod)
      implicit real*8 (a-h,o-z)
c
c     ep      equivalent plastic strain
c     sy      yield stress
c     h       slope in terms of equivalent stress /
c                                   equivalent plastic strain
c     temp    temperature
c

     (user coding to be included here.)

      return
      end
```

where:

EBARPL is the current total equivalent plastic strain,

$$\bar{\varepsilon}^p = \sum d\bar{\varepsilon}^p$$

$$d\bar{\varepsilon}^p = \sqrt{\frac{2}{3}\, d\varepsilon^p_{ij}\, d\varepsilon^p_{ij}}$$

HSLOPE is the work-hardening slope, defined as:

$$\frac{d\bar{\sigma}}{d\bar{\varepsilon}^p}$$

SHIFT is the amount of origin shift in the kinematic or combined hardening models.

SYIELD is the current yield stress $\bar{\sigma}$

TEMP is the current temperature.

ININOD is the current node number.

NOTE: This is not the slope of the tensile stress-strain curve, which is $\frac{d\bar{\sigma}}{d\bar{\varepsilon}}$ with:

$$d\bar{\varepsilon} = d\bar{\varepsilon}^{e} + d\bar{\varepsilon}^{p}$$

The user must take care to provide rate of change of stress with respect to plastic strain, NOT total strain.

The user must define HSLOPE and SYIELD in this routine: EBARPL, TEMP and ININOD should not be changed. The call to this subroutine is invoked by not specifying hardening slope at the *WORKHARD data section. Note that the first yield stress must be specified in the input data deck. The routine is called as required by the program during elastic-plastic calculations. The number of times it is called per increment depends on the number of points going plastic, the nonlinearity of the work-hardening curve, and on temperature dependence.

F.5 SUBROUTINE UCOEF

The default for thermal expansion effects is isotropic. If anisotropic coefficients are desired, the user subroutine UCOEF can be used. UCOEF is coded and added to the system as follows:

```
c=subroutine=ucoef  called by 'thrstn'
      subroutine ucoef(coef,nelstr,ndi,nshear,node,temp)
c
      implicit real*8( a- h , o - z )
c
      dimension coef(1)
c
c     coef  anisotropic coefficient of thermal expansion
c     nelstr  number of strain components
c     ndi     number of direct strain components
c     nshear  number of shear strain components
c     node    node number
c     temp    current temperature
c
      (user coding to be included here)

      return
      end
```

F.6 SUBROUTINE ANPLAS

The default in the MHOST code for the initial yield surface is isotropic. An anisotropic yield surface option can be obtained through the ANPLAS user subroutine.

The anisotropic yield function and stress potential are assumed as:

$$a_1(\sigma_y-\sigma_z)^2+ a_2(\sigma_z-\sigma_x)^2+ a_3(\sigma_x-\sigma_y)^2+ 3a_4\tau_{yz}^2 + 3a_5\tau_{zx}^2 + 3a_6\tau_{xy}^2 = 2\bar{\sigma}^2$$

(R. Hill - Mathematical Theory of Plasticity, Oxford, 1950)

where

is the equivalent tensile yield stress for isotropic behavior.

The user defines ratios of actual to isotropic yield (in the preferred orientation) in the array YRDIR for direct tension yielding, and YRSHR for yield in shear (ratio of actual shear yield to / 3, where s is the isotropic yield stress in sure shear. Then the a_i's above are derived as (Hill):

$$a_1 = \frac{1}{\text{YRDIR(2)**2}} + \frac{1}{\text{YRDIR(3)**2}} - \frac{1}{\text{YRDIR(1)**2}}$$

$$a_2 = \frac{1}{\text{YRDIR(3)**2}} + \frac{1}{\text{YRDIR(1)**2}} - \frac{1}{\text{YRDIR(2)**2}}$$

$$a_3 = \frac{1}{\text{YRDIR(1)**2}} + \frac{1}{\text{YRDIR(2)**2}} - \frac{1}{\text{YRDIR(3)**2}}$$

$$a_4 = \frac{2}{\text{YRSHR(3)**2}}$$

$$a_5 = \frac{2}{\text{YRSHR(2)**2}}$$

$$a_6 = \frac{2}{\text{YRSHR(1)**2}}$$

```
      subroutine anplas(intnod,jlaw,ndi,nshear,yrdir,yrshr)
c
c ************************************************************************
c
      implicit real*8 ( a-h , o-z )
               real*4   rwork
c
c ************************************************************************
```

```
c
      dimension yrdir(3),yrshr(3)
c
c     n       node number
c     ndi     number of direct stress components
c     nshear  number of shear stress components
c     yrdir   direct ratios
c     yrshr   shear ratios
c

       (user coding to be included here)

      return
      end
```

F.7 SUBROUTINE CRPLAW

Two types of conventional creep strain laws may be provided by the user. The first through user subroutine CRPLAW results in deviatoric creep strain; the second through user subroutine VSWELL results in dilatation (volumetric) creep strain. These routines will be called for each nodal point.

The program allows the user to input a general creep law through subroutine CRPLAW.

The assumed form of the law is:

$$e = f(\bar{\sigma}, T, t, \bar{\varepsilon}^c, p, \alpha_1, \alpha_2, \text{etc.})$$

where:

$\dot{\bar{\varepsilon}}^c$ is the equivalent creep strain rate, in uniaxial tension.

$\bar{\sigma}$ is the current equivalent (J_2) stress, normalized for uniaxial tension.

T is the current total temperature.

t is the current total time.

$\bar{\varepsilon}^c$ is the current total equivalent creep strain, normalized for uniaxial tension.

p is the hydrostatic stress.

$\alpha_1, \alpha_2, \ldots$ are additional parameters to be employed at the user discretion.

The program requires the user to program a creep law so that an equivalent creep strain increment is defined. An example is given below.

Subroutine CRPLAW is written with the following header records:

```
      subroutine crplaw(eqcpi,eqprev,dum,temp,timinc,totinc,intnod)
c
c ***********************************************************************
c
      implicit real*8 ( a-h , o-z )
               real*4   rwork
c
```

```
c ************************************************************************
c
      dimension dum(3)

       (user coding to be included here)

      return
      end
```

The simplest way to define a creep strain increment from a given rate law $\dot{e}^c = f(\bar{\sigma}$, etc) is to multiply by t so that $\Delta\varepsilon^c = \Delta t\, f(\bar{\sigma}, \text{etc.})$.

F.8 SUBROUTINE VSWELL

The user subroutine VSWELL allows the user to include pure swelling (dilatational) creep in MHOST.

Subroutine VSWELL is written with the following header records:

```
      subroutine vswell(swell,dss,temp,timinc,totinc,intnod)
      implicit real*8 (a-h,o-z)
c
      dimension dss( 3 )
c
c     swell   is the incremental swelling strain
c     dss(1)  is the equivalent stress
c     dss(2)  is the dilatational stress
c     dss(3)  is the volumentric creep strain
c     temp    is the total temperature
c     timinc  is the size of the current time increment
c     totinc  is the total time at the beginning of the increment
c     intnod  is the node number
c

       (user coding to be included here)

      return
      end
```

F.9 SUBROUTINE WALCON

The temperature dependent material constants for the Walker creep plasticity model are calculated in the user subroutine WALCON. An example used for the calculation of demonstration problem A.4 is given as follows:

```
      subroutine walcon(indep,itherm,tempm,
     1 ee,anu,ak1,ak2,anin,am,an1,an2,an3,an4,an5,an6,an7,
     2 omeg0,akind,siinf,dn1dt,dn2dt,domdt,
     3 an,alam,amu,c1,c2,c3,c4,c5)
      implicit real*8 (a-h,o-z)
c     this subroutine is called by hypela to calculate all of the
c     temperature dependent material constants
c     except for a few changes of names, this routine is as delivered
c     by b. cassenti u.t. research
      dimension tabt(6),eet(6),anut(6),ak1t(6),anint(6),amt(6),an1t(6)
      dimension an2t(6),an3t(6),an4t(6),an5t(6),an6t(6),an7t(6),ak2t(6)
      dimension omeg0t(6)
      dimension akindt(6),siinft(6)
      data tabt/800.d0,1000.d0,1200.d0,1400.d0,1600.d0,1800.d0/
      data eet/26.d6,24.d6,24.d6,22.6d6,18.6d6,13.2d6/
      data anut/0.322d0,0.328d0,0.334d0,0.339d0,0.345d0,0.351d0/
      data ak1t/50931.d0,75631.d0,95631.d0,110696.d0,91505.d0,59292.d0/
      data ak2t/0.d0,0.d0,0.d0,0.d0,0.d0,0.d0/
      data anint/.059d0,.059d0,.079d0,.1497d0,.195d0,.223d0/
      data amt/1.158d0,1.158d0,1.158d0,1.158d0,1.158d0,1.158d0/
      data an1t/0.d0,0.d0,0.d0,0.d0,0.d0,0.d0/
      data an2t/1.d7,1.9d7,1.5d7,2.d7,5.d6,1.d6/
      data an3t/250.d0,320.d0,781.2d0,1178.6d0,672.6d0,312.5d0/
      data an4t/0.d0,0.d0,0.d0,0.d0,0.d0,0.d0/
      data an5t/0.d0,0.d0,0.d0,0.d0,0.d0,0.d0/
      data an6t/0.d0,0.d0,0.d0,0.d0,8.977d-4,2.733d-3/
      data an7t/0.d0,0.d0,0.d0,0.d0,0.d0,0.d0/
      data omeg0t/0.d0,0.d0,-2000.d0,-2000.d0,-1434.d0,-1200.d0/
      data akindt/0.d0,0.d0,0.d0,0.d0,0.d0,0.d0/
      data siinft/48.d3,48.d3,60.d3,1.d10,1.d10,1.d10/
      ntp=6
      ntpm1=ntp-1
      tdif=tabt(2)-tabt(1)
      l1=tempm
      l2=tabt(1)-tdif
      l3=tdif
      it=(l1-l2)/l3
      if(it.lt.1)it=1
      if(it.gt.ntpm1)it=ntpm1
      fac=(tempm-tabt(it))/tdif
      ee=(eet(it+1)-eet(it))*fac+eet(it)
      anu=(anut(it+1)-anut(it))*fac+anut(it)
      ak1=(ak1t(it+1)-ak1t(it))*fac+ak1t(it)
```

```
      ak2=(ak2t(it+1)-ak2t(it))*fac+ak2t(it)
      anin=(anint(it+1)-anint(it))*fac+anint(it)
      am=(amt(it+1)-amt(it))*fac+amt(it)
      an1=(an1t(it+1)-an1t(it))*fac+an1t(it)
      an2=(an2t(it+1)-an2t(it))*fac+an2t(it)
      an3=(an3t(it+1)-an3t(it))*fac+an3t(it)
      an4=(an4t(it+1)-an4t(it))*fac+an4t(it)
      an5=(an5t(it+1)-an5t(it))*fac+an5t(it)
      an6=(an6t(it+1)-an6t(it))*fac+an6t(it)
      an7=(an7t(it+1)-an7t(it))*fac+an7t(it)
      omeg0=(omeg0t(it+1)-omeg0t(it))*fac+omeg0t(it)
      if (indep .eq. 0) goto 65
      akind=(akindt(it+1)-akindt(it))*fac+akindt(it)
      siinf=(siinft(it+1)-siinft(it))*fac+siinft(it)
   65 if (itherm .eq. 0) goto 73
      dn1dt=(an1t(it+1)-an1t(it))/tdif/(an1+1.d-3)
      dn2dt=(an2t(it+1)-an2t(it))/tdif/(an2+1.d-3)
      domdt=(omeg0t(it+1)-omeg0t(it))/tdif/(omeg0+1.d-3)
   73 continue
      an=1./anin
      alam=ee*anu/((1.d0-2.d0*anu)*(1.d0+anu))
      amu=(1.d0-2.d0*anu)*alam/(2.d0*anu)
      c1=2.d0*amu*alam/(alam+2.d0*amu)
      c2=4.d0*amu*(alam+amu)/(alam+2.d0*amu)
      c3=2.d0*amu
      c4=amu
      c5=alam
      return
      end
```

Details of the Walker creep plasticity model can be found in B. N. Cassenti, Research and Development Program for the Development of Advanced Time-Temperature Dependent Constitutes Relationships, NASA CR-168191 (1983), Appendix 2.

F.10 SUBROUTINE UBOUN

UBOUN generates nodal displacement constraints.

Arguments are:

NODE (NKBC):	Array for the index of constrained nodal points
NDOF (NKBC):	Array for the index of constrained degrees-of-freedom
DISP (NKBC):	Constrained displacement (output)
NKBC :	Number of displacement constraint
RUNTIM :	Total time
TIMINC :	Current time increment
JINC :	Current increment number

In the current MHOST installation, the following example is included:

```
      subroutine uboun(node,ndof,disp,nkbc,runtim,timinc,jinc)
      implicit real*8 (a-h,o-z)
      dimension node(nkbc),ndof(nkbc),disp(nkbc)
      fac=1.570796327d0
      t1=runtim*fac/15.d0
      t2=t1+timinc*fac/15.d0
      disp2=-.00125*dsin(t2)
      disp1=-.00125*dsin(t1)
      dispin=disp2-disp1
      do 1 i=1,nkbc
      n1=node(i)
      n2=ndof(i)
      if(n1.eq.2.and.n2.eq.1) disp(i)=dispin
      if(n1.eq.3.and.n2.eq.1) disp(i)=dispin
    1 continue
      return
      end
```

F.11 FUNCTION USXX

The user-definable function USXX allows the user to specify problem-specific spatial correlation functions for load spectra to be used in frequency-domain analysis of random vibration problems. A listing of the function is provided below.

```
c ... function usxx ... called from subroutine 'intsqq'
c
      function usxx( ni, nj, coor, nnode, maxcrd, omega )
c
c ********************************************************************
c
c     user-defined correlation function between the fourier spectra
c     of the random loading between ni and nj at frequency omega
c
c     input
c     -----
c     ni      :  node number i
c     nj      :  node number j
c     coor    :  node coordinates  ( x, y, z... )
c     nnode   :  number of nodes in the mesh
c     maxcrd  :  maximum number of coordinates per node
c     omega   :  frequency of excitation
c
c     output
c     ------
c     usxx    :  correlation value for the load spectra
c
c     usage
c     -----
c     (1)  for full correlation, use function as it is, with usxx = 1
c     (2)  for no correlation, remove 'c#' to get usxx = 1 if ni = nj,
c          and usxx = 0 otherwise
c     (3)  for partial correlation, the appropriate correlation function
c          must be coded up by the user
c
c ********************************************************************
c
      implicit real*8 (a-h,o-z)
c
      dimension coor(maxcrd,nnode)
c
      usxx = 1.0d0
c#    if ( ni .ne. nj ) usxx = 0.0d0
c
      return
      end
```

G. ELEMENT LIBRARY

The following subsection is a quick reference for the elements which are contained in the current version of the MHOST program. The basic characterization of the elements is given by the parameters in the following table.

TABLE G.1 ELEMENT TYPE CHARACTERIZATION TABLE

ELEMENT	TYPE	NELCRD	NELNFR	NELNOD	NELSTR	NELCHR	NELINT	NELLV	NELLAY	NELCMP	NDI	NSHEAR	JLAW
PLANE STRESS	3	2	2	4	3	6	4	8	1	3	2	1	2
PLANE STRAIN	11	2	2	4	4	6	4	8	1	4	3	1	3
AXISYMMETRIC	10	2	2	4	4	6	4	8	1	4	3	1	4
3D SOLID	7	3	3	8	6	6	8	24	1	6	3	3	5
4-NODE SHELL	75	7	6	4	9	6	4	24	5	5	2	3	6
2-NODE BEAM	98	6	6	2	6	6	1	12	1	6	3	3	7
PLANE STRESS*	101	2	2	9	3	6	9	18	1	3	2	1	2
PLANE STRAIN*	102	2	2	9	4	6	9	18	1	4	3	1	3
AXISYMMETRIC*	103	2	2	9	4	6	9	18	1	4	3	1	4
3D SOLID*	104	3	3	27	6	6	27	81	1	6	3	3	5
PLANE STRESS	151	2	2	4	3	6	4	8	1	3	2	1	2
PLANE STRAIN	152	2	2	4	4	6	4	8	1	4	3	1	3
AXISYMMETRIC	153	2	2	4	4	6	4	8	1	4	3	1	4
3D SOLID	154	3	3	8	6	6	8	24	1	6	3	3	5

DEFINITION OF ELEMENT PARAMETERS:

NELCRD	Number of coordinates per node
NELNFR	Number of degrees-of-freedom per node
NELNOD	Number of nodes per element
NELSTR	Number of generalized stress and/or strain components
NELCHR	Number of element material properties
NELINT	Number of 'full' integration points per element
NELLV	Total number of degrees-of-freedom per element
NELLAY	Number of integration layers through the thickness
NELCMP	Number of stress and/or strain components at a point
NDI	Number of direct stress components at a point
NSHEAR	Number of shear stress components at a point
JLAW	Type of constitutive model for D-matrix formulation

*NOTE: Element types 101 through 104 are available only within the context of the subelement mesh refinement option. No details will be given in this section. Not recommended for the use in the conventional analysis.

FOUR-NODE PLANE STRESS ELEMENT - MHOST ELEMENT TYPE 3

I. SUMMARY

This element is a four-noded isoparametric plane stress element involving two degrees-of-freedom per node as shown in Figure G.1. The degrees-of-freedom are

$\underline{u} = [\ u_x, u_y\ ]^T$

in the rectangular Cartesian coordinate system and are identified by the integer numbers 1 and 2.

Three strain components are defined at the nodes and the element integration points. These are

$\underline{\varepsilon} = [\ \varepsilon_x, \varepsilon_y, \gamma_{xy}\ ]^T$

and the stress vector is defined as:

$\underline{\sigma} = [\ \sigma_x, \sigma_y, \tau_{xy}\ ]^T$

The definition of stresses and strains follows standard engineering convention.

II. DATA PREPARATION

The particular data lines related to the Element Type 3 are described below.

1. PARAMETER DATA

*ELEMENT — Followed by an integer parameter giving the upper bound to the number of elements. After the keyword line, the number 3 is specified as an integer parameter.

*DISTRIBUTEDLOAD
Used, without a parameter, when the distributed loading option is invoked.

2. MODEL DATA

*DISTRIBUTEDLOAD — One parameter specifying element type (3). Subsequent data lines contain three integers and two reals.

- Integer 1: The first element of the series subjected to the distributed loading.
- Integer 2: The last element of the series.
- Integer 3: Pressure loading type. Value one (1) for traction along the element edge 1 and 2.
- REAL 1: Intensity of normal surface traction force (positive inward) as shown in Figure G.1.
- REAL 2: Intensity of body force loading. The total body force is this value times the absolute value of the vector defining the direction of the body force. In the case of centrifugal loading, the angular velocity (radians/time) should be input.

*PROPERTIES — One parameter specifying element type (3). Subsequent data lines contain two integers and five reals.

- Integer 1: The first node of the series to which the current matrial data applies.
- Integer 2: The last node of the series.
- REAL 1: The thickness of the material
- REAL 2: Young's modulus

REAL 3: Poisson's ratio

REAL 4: Thermal expansion coefficient

REAL 5: Density

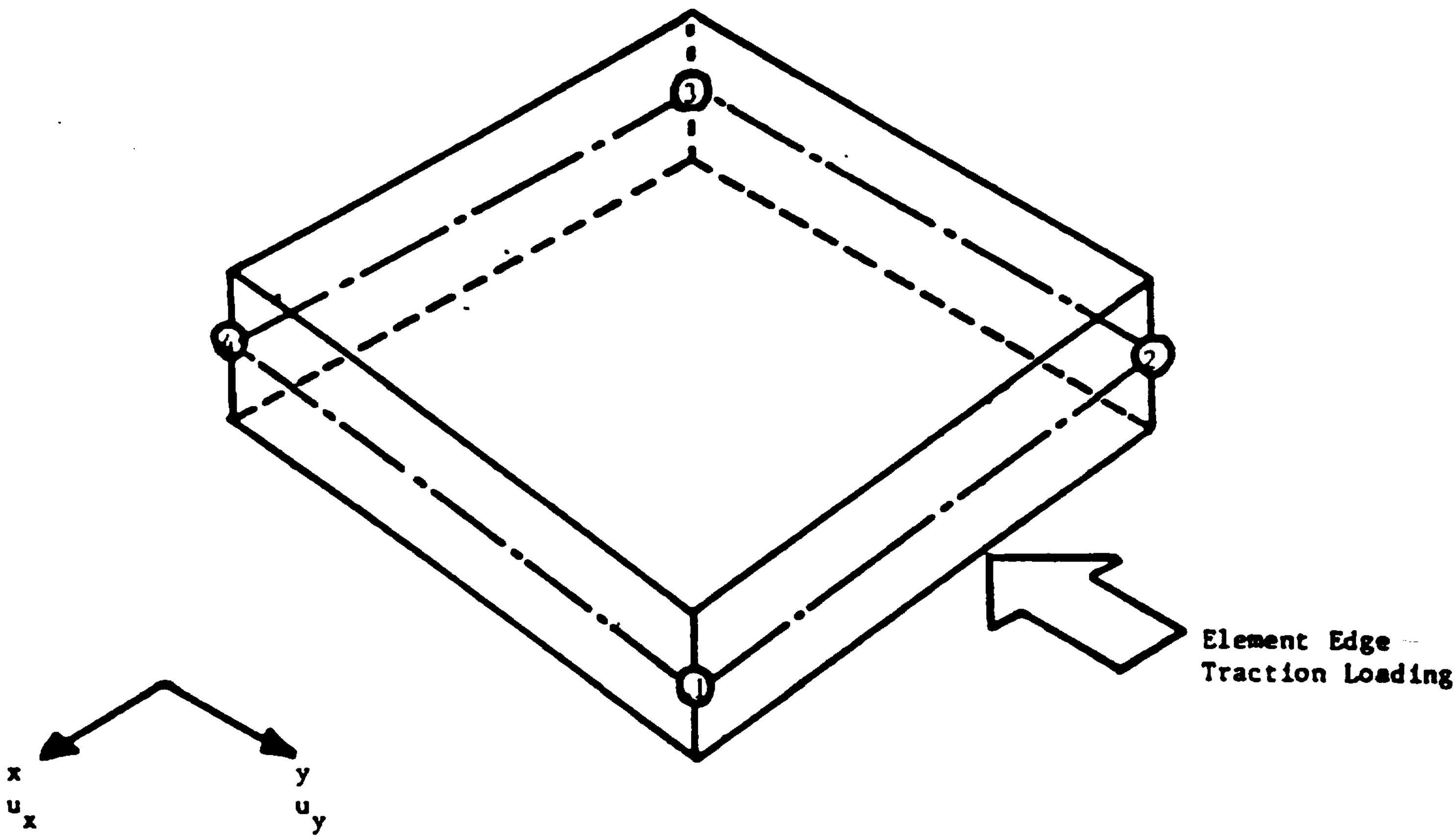

FIGURE G.1 Plane Stress Elements Traction Loading Definition

EIGHT-NODE THREE-DIMENSIONAL SOLID ELEMENT - MHOST ELEMENT TYPE 7

I. SUMMARY

This element is a eight-noded isoparametric solid element with three degrees-of-freedom per node as shown in Figure G.2. The degrees-of-freedom are

$$\underline{u} = [\, u_x, u_y, u_z \,]^T$$

in the rectangular Cartesian coordinate system and are identified by the integer numbers 1 through 3 respectively.

Six strain components are defined at the nodes and the element integration points. These are

$$\underline{\varepsilon} = [\, \varepsilon_x, \varepsilon_y, \varepsilon_x, \gamma_{xy}, \gamma_{yz}, \gamma_{zx} \,]^T$$

and the stress vector is defined as

$$\underline{\sigma} = [\, \sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx} \,]^T$$

The definition of stresses and strains follows standard engineering convention.

II. DATA PREPARATION

The particular data lines for Element Type 7 are described below.

1. PARAMETER DATA

*ELEMENT — Followed by an integer parameter giving the upper bound to the number of elements. After the keyword line, the number 7 is specified as an integer parameter.

*DISTRIBUTEDLOAD
Used, without a parameter, when the distributed loading option is invoked.

2. MODEL DATA

*DISTRIBUTEDLOAD — One parameter specifying element type (7). Subsequent data lines contain three integers and two reals.

Integer 1: The first element of the series subjected to the distributed loading.

Integer 2: The last element of the series.

Integer 3: Pressure loading type. Value one (1) for traction along the element surface 1,2,3 and 4.

REAL 1: Intensity of normal surface traction force (positive inward) as shown in Figure G.2.

REAL 2: Intensity of body force loading. The total body force is this value times the absolute value of the vector defining the direction of the body force. In the case of centrifugal loading, the angular velocity (radians/time) should be input.

*PROPERTIES — One parameter specifying element type (7). Subsequent data lines contain two integers and five reals.

Integer 1: The first node of the series to which the current matrial data applies.

Integer 2: The last node of the series.

REAL 1: Dummy entry

REAL 2: Young's modulus

REAL 3: Poisson's ratio

REAL 4: Thermal expansion coefficient

REAL 5: Density

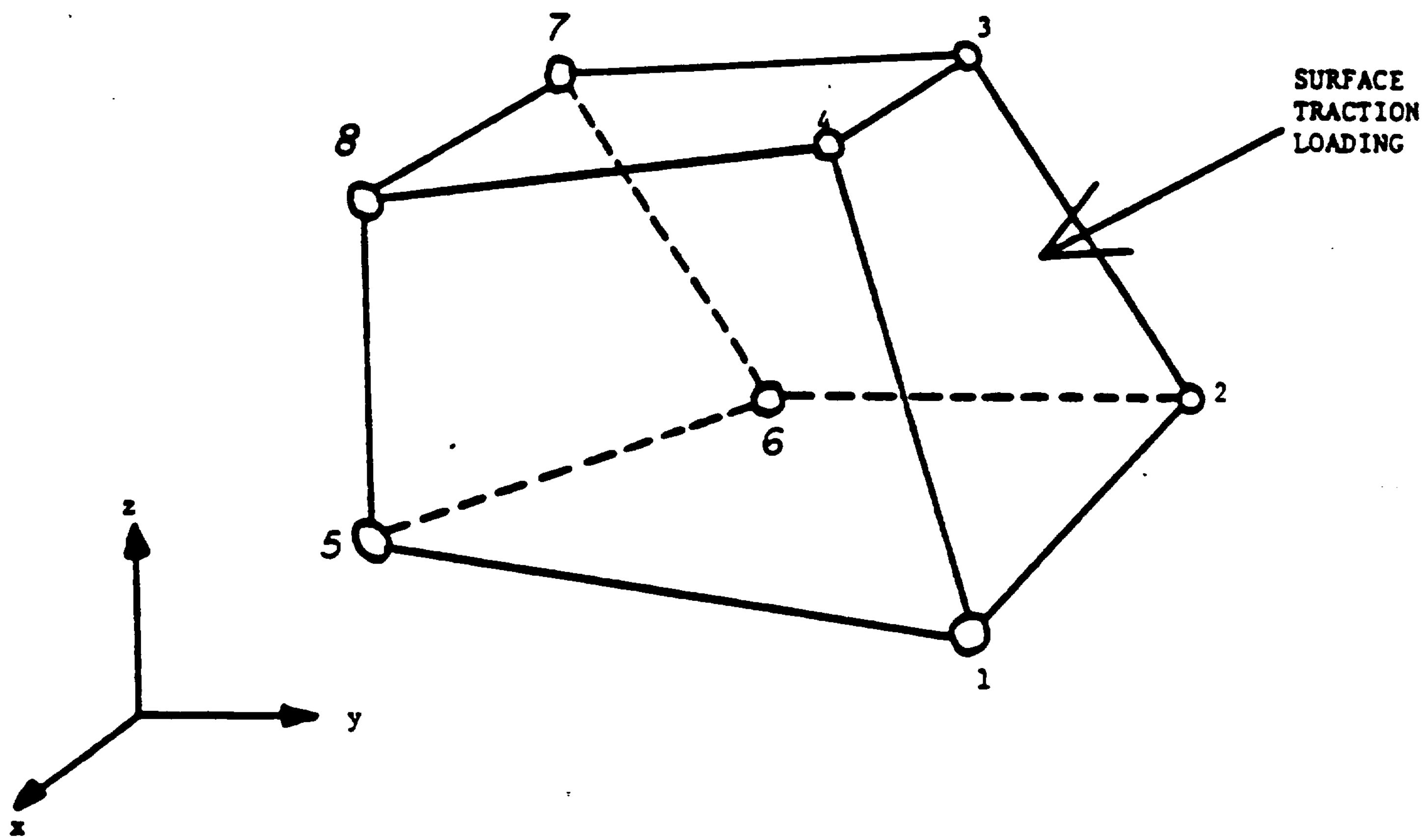

FIGURE G.2 Solid Element Traction Loading Definition

FOUR-NODE AXISYMMETRIC SOLID ELEMENT - MHOST ELEMENT TYPE 10

I. SUMMARY

This element is a four-noded isoparametric axisymmetric solid element involving two degrees-of-freedom per node as shown in Figure G.3. The degrees-of-freedom are

$$\underline{u} = [\ u_z,\ u_r\ ]^T$$

in the cylindrical coordinate system and are identified by the integer numbers 1 and 2, respectively.

Four strain components are defined at the nodes and the element integration points. These are

$$\underline{\varepsilon} = [\ \varepsilon_z, \varepsilon_r, \varepsilon_\theta, \gamma_{zr}\ ]^T$$

and the stress vector is defined as

$$\underline{\sigma} = [\ \sigma_z, \sigma_r, \sigma_\theta, \tau_{zr}\ ]^T$$

The definition of stresses and strains follows standard engineering convention.

II. DATA PREPARATION

The particular data lines for Element Type 10 are described below.

1. PARAMETER DATA

*ELEMENT — Followed by an integer parameter giving the upper bound to the number of elements. After the keyword line, the number 10 is specified as an integer parameter.

*DISTTRIBUTEDLOAD
Used without a parameter, when the distributed loading option is invoked.

2. MODEL DATA

*DISTRIBUTEDLOAD — One parameter specifying element type (10). Subsequent data lines contain three integers and two reals.

Integer 1: The first element of the series subjected to the distributed loading.

Integer 2: The last element of the series.

Integer 3: Pressure loading type. Value one (1) for traction along the element edge 1 and 2.

REAL 1: Intensity of normal surface traction force (positive inward) as shown in Figure G.3.

REAL 2: Intensity of body force loading. The total body force is this value times the absolute value of the vector defining the direction of the body force. In the case of centrifugal loading, the angular velocity (radians/time) should be input.

The gravity force can only be applied in the axial (z) direction, and the centrifugal force can only be imposed around the z axis.

*PROPERTIES — One parameter specifying element type (10). Subsequent data lines contain two integers and five reals.

Integer 1: The first node of the series to which the current matrial data applies.

Integer 2: The last node of the series.

 C-2

REAL 1:	Dummy entry
REAL 2:	Young's modulus
REAL 3:	Poisson's ratio
REAL 4:	Thermal expansion coefficient
REAL 5:	Density

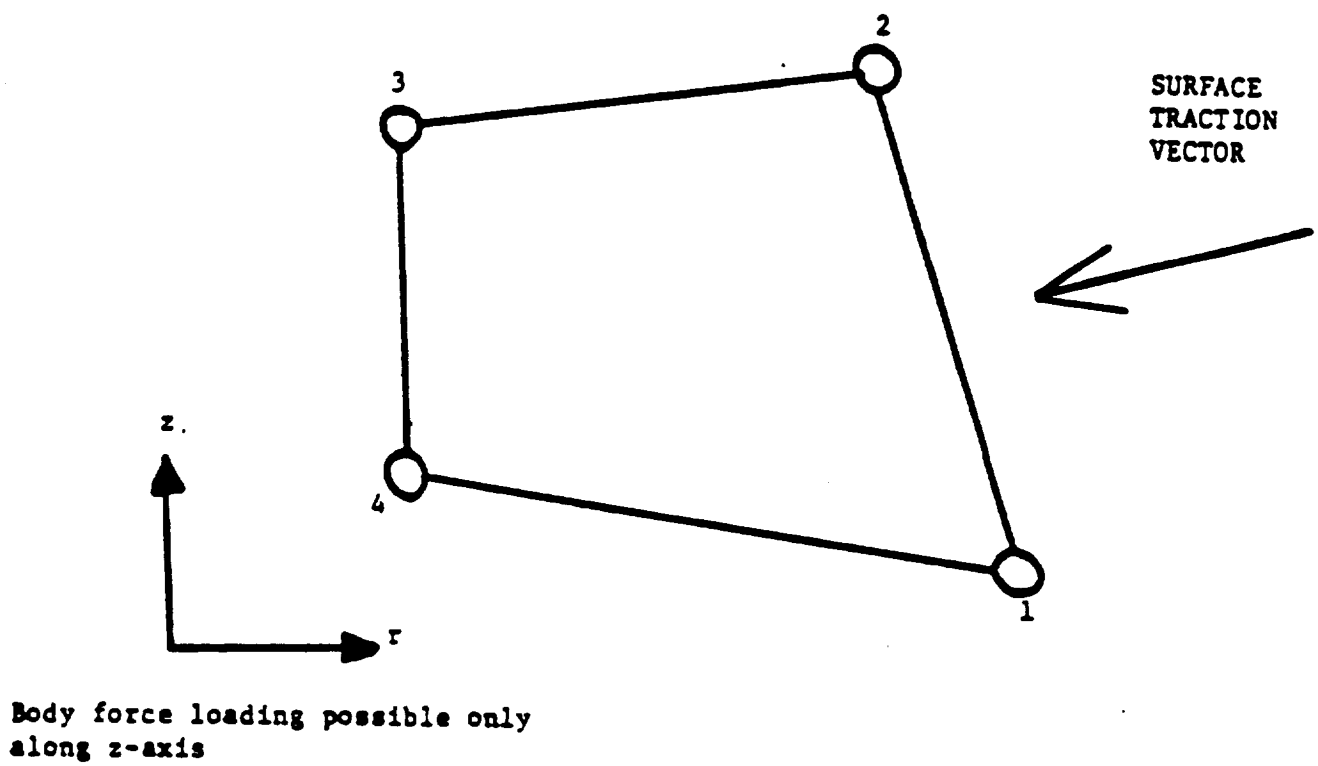

FIGURE G.3 Axisymmetric Solid Element Traction Loading Definition

FOUR-NODE PLANE STRAIN ELEMENT - MHOST ELEMENT TYPE 11

I. SUMMARY

This element is a four-noded isoparametric plane strain element involving two degrees-of-freedom per node as shown in Figure G.4. The degrees-of-freedom are

$$\underline{u} = [\ u_x,\ u_y\ ]^T$$

in the rectangular Cartesian coordinate system and are identified by the integer numbers 1 and 2, respectively.

Four strain components are defined at the nodes and the element integration points. These are

$$\underline{\varepsilon} = [\ \varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}\ ]^T$$

and the stress vector is defined as

$$\underline{\sigma} = [\ \sigma_x, \sigma_y, \sigma_z, \tau_{xy}\ ]^T$$

The definition of stresses and strains follows standard engineering convention.

II. DATA PREPARATION

The particular data lines related to the Element Type 11 are described below.

1. PARAMETER DATA

*ELEMENT Followed by an integer parameter giving the upper bound to the number of elements. After the keyword line, the number 11 is specified as an integer parameter.

*DISTRIBUTEDLOAD
Used, without a parameter, when the distributed loading option is invoked.

2. MODEL DATA

*DISTRIBUTEDLOAD One parameter specifying element type (11). Subsequent data lines contain three integers and two reals.

Integer 1: The first element of the series subjected to the distributed loading.

Integer 2: The last element of the series.

Integer 3: Pressure loading type. Value one (1) for traction along the element edge 1 and 2.

REAL 1: Intensity of normal surface traction force (positive inward).

REAL 2: Intensity of body force loading. The total body force is this value times the absolute value of the vector defining the direction of body force. In the case of centrifugal loading, the angular velocity (radians/time) should be input.

*PROPERTIES One parameter specifying element type (11). Subsequent data lines contain two integers and five reals.

Integer 1: The first node of the series to which the current matrial data applies.

Integer 2: The last node of the series.

REAL 1: Dummy entry

REAL 2: Young's modulus

REAL 3: Poisson's ratio

REAL 4: Thermal expansion coefficient

REAL 5: Density

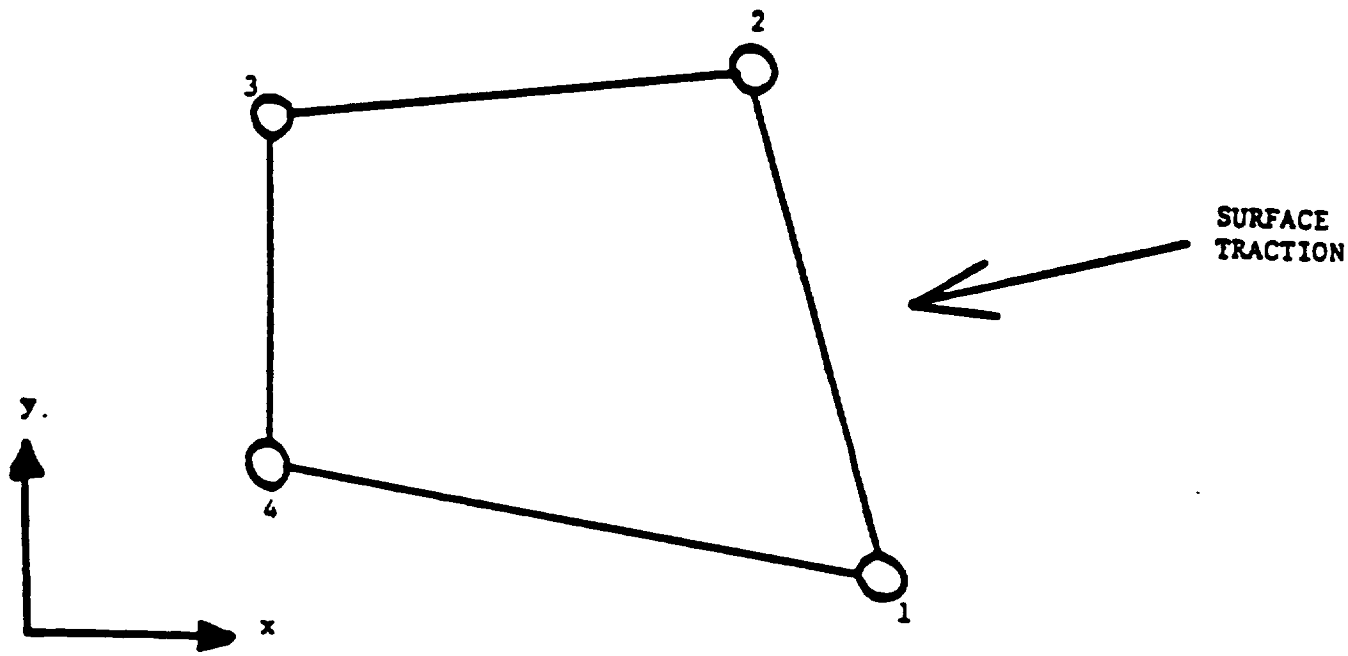

FIGURE G.4 Plane Strain Element Traction Loading Definition

FOUR-NODE SHELL ELEMENT - MHOST ELEMENT TYPE 75

I. SUMMARY

This element is a four-node isoparametric shell element derived from the Reissner-Mindlin theory for plates and shells, involving six degrees-of-freedom per node. The degrees-of-freedom are

$$\underline{u} = [\ u_x,\ u_y,\ u_z, \theta_x, \theta_y, \theta_z]^T$$

and are identified by an integer numbers 1 through 6 respectively. See Figure G.5 for the geometrical definition. The characteristics of this element are similar to MARC Element 75.

Eight generalized strain components are defined at the nodes with respect to the local Cartesian coordinate system. These are

$$\underline{\varepsilon} = [\varepsilon_x, \varepsilon_y, \gamma_{xy}, \gamma_{xz}, \gamma_{yz}, \kappa_x, \kappa_y, \kappa_{xy}]^T$$

and the generalized stress components associated with these entries are

$$\underline{\sigma} = [N_x,\ N_y,\ N_{xy},\ S_{xz},\ S_{yz},\ M_x,\ M_y,\ M_{xy}]^T$$

An orthogonal coordinate system is constructed at the element centroid and is used to define the eight strain components.

$$\underline{\varepsilon}' = [\varepsilon_x', \varepsilon_y', \varepsilon_{xy}', \psi_{xz}', \psi_{yz}', \kappa_x', \kappa_y', \kappa_{xy}']^T$$

The first three components are membrane strains, the fourth and fifth components are transverse shear strains, and the last three components are curvature strains. These components are used to form the element stiffness matrix.

The displacements and stresses are read in and printed out in terms of nodal Cartesian components. The only exception occurs when the element quantity output routines are invoked with the PRINTOPTION command. In this case information is given in the local element orthogonal coordinate system.

The local coordinate system at a node is defined by calculating the element normal vector for all the elements connected to the nodal point and the average is taken to generate the normal-to-the-shell-surface vector at the node. Components of this vector are printed before reporting the results of the zeroth increment. The tangential coordinate system is calculated by taking the cross products of the nodal normal to global y-axis for the local x-axis, and the global x-axis for the local y-axis, respectively. Algebraic details can be found in O. C.

Zienkiewicz, The Finite Element Method, 3rd Ed., McGraw-Hill, New York (1977).

II. DATA PREPARATION

The particular input data lines related to Element Type 75 are described here. For further detail, please refer to Sections B through E.

1. PARAMETER DATA

*ELEMENT — Followed by an integer parameter specifying an upper bound for the number of elements. After the keyword line, the number 75 is specified as an integer parameter.

*DISTRIBUTEDLOAD — Used, without a parameter, when the distributed loading is specified.

*LOUBIGNAC — The third parameter associated with the stiffness matrix integration is not applicable to this element.

2. MODEL DATA

*COORDINATES — Subsequent data records contain one integer to identify the nodal point, and three global coordinates for the particular node. In case of flat plates in the x-y plane, the thin coordinate can be omitted. The fourth real data item is the shell thickness at the node.

*DISTRIBUTEDLOAD — One parameter of element type (75). Subsequent data lines contain three integers and two real parameters. These are

Integer 1: First element in a consecutive series of elements.

Integer 2: Last element in the same consecutive series of elements.

Integer 3: 1 for the edge traction loading.
10 for the lateral pressure loading.
11 for the both.

REAL 1: Value for the uniform edge traction.

REAL 2: Value for the lateral pressure loading.

REAL 3: Intensity of the body force loading. The total body force is this value times the absolute value of the vector defining the direction of body force. In the case of centrifugal loading, the angular velocity (radians/time) should be input.

*PROPERTIES	With one parameter specifying element type (75), subsequent data records contain two integer and three real parameters.
Integer 1:	First node in a consecutive series of nodes.
Integer 2:	Last node in the same consecutive series of nodes.
REAL 1:	Thickness of the shell.
REAL 2:	Young's modulus.
REAL 3:	Poisson's ratio.
REAL 4:	Coefficient of thermal expansion.
REAL 5:	Mass density.

III. LINE PRINTER OUTPUT

Stresses and strains are printed at each layer of integration through the thickness. Components printed are:

1. σ_x
2. σ_y
3. τ_{xy}
4. τ_{xz}
5. τ_{yz}

The default number of layers is five, with the first and the last layers being the bottom and the top surfaces of the shell, respectively. The rest of the layers are uniformly distributed through the thickness.

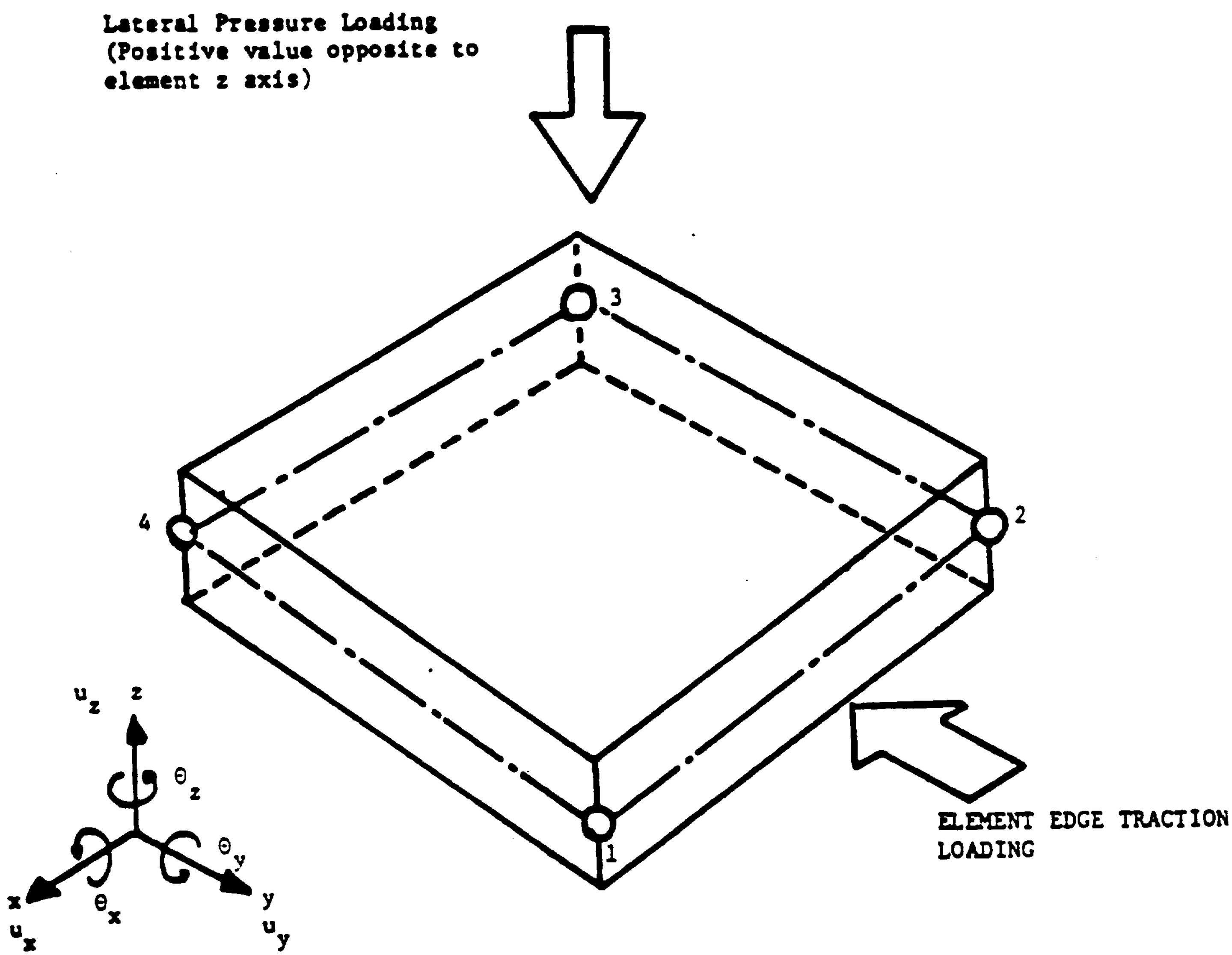

FIGURE G.5 Shell Element Traction Loading Definitions

TWO-NODE TIMOSHENKO BEAM ELEMENT - MHOST ELEMENT TYPE 98

I. SUMMARY

This element is a two-node linear isoparametric beam element derived from Timoshenko beam theory, involving six degrees-of-freedom per node. The degrees-of-freedom are

$$\underline{u} = [u_x, u_y, u_z, \theta_x, \theta_y, \theta_z]^T$$

and are identified by integers from 1 through 6, respectively. See Figure G.6 for geometrical definition of this element. The characteristics of this element are similar to MARC Element 98.

Six generalized strain components are defined at the nodes with respect to the local Cartesian coordinate system defined at the nodes. The six generalized strain components are

$$\underline{\varepsilon} = [\gamma_{xz}, \gamma_{yz}, \varepsilon_z, \kappa_x, \kappa_y, \kappa_z]^T$$

and the corresponding generalized stress components are

$$\underline{\sigma} = [F_x, F_y, F_z, M_x, M_y, M_z]^T$$

At the element's single integration point, an orthogonal coordinate system is defined, having the z-axis aligned with the element axis and the y-axis in the plane defined by the element axis and the average of the normals at the two end nodes. The element stiffness matrix is derived in terms of the eight generalized strain components at this point, namely

$$\underline{\varepsilon}' = [\gamma'_{xz}, \gamma'_{yz}, \varepsilon'_z, \kappa'_x, \kappa'_y, \psi'_z]^T$$

with the first two components associated with transverse shear, the third with axial elongation, the fourth and fifth with transverse curvature, and the last one with axial twist.

II. DATA PREPARATION

1. PARAMETER DATA

*ELEMENT — Followed by an integer parameter with upper bound to the number of elements. After the keyword line, the number 98 must be specified as an the integer parameter.

*BEAMSECTION — Followed by an integer parameter with upper bound to the number of different beam sections specified.

*DISTRIBUTEDLOAD — Used, without a parameter, when distributed loadings are specified.

2. MODEL DATA

*COORDINATES — Subsequent data records contain one integer to identify the node number, followed by the three global coordinates of the node and three components of the normal vector (not necessarily a unit vector) used to define the plane of the local beam y-axis at the node.

*BEAMSECTION — Subsequent data records contain two integers and six real parameters, specifying:

Integer 1: First node in a consecutive series of nodes.

Integer 2: Last node in the same consecutive series of nodes.

REAL 1: Shear Area in the local x-direction.

REAL 2: Shear Area in the local y-direction.

REAL 3: Cross-Sectional Area.

REAL 4: Moment of Inertia about the local x-axis.

REAL 5: Moment of Inertia about the local y-axis.

REAL 6: Torsional Constant.

*DISTRIBUTEDLOAD — Followed by one integer parameter with element type (98). Subsequent data lines contain three integers and three real parameters, specifying.

Integer 1: First element in a consecutive series of elements.

Integer 2: Last element in the same consecutive series of elements.

Integer 3: Dummy parameter, any positive integer.

REAL 1: Line load in the local x-direction, positive as shown in Figure G.6.

REAL 2: Line load in the local y-direction, positive as shown in Figure G.6.

REAL 3: Intensity of body force loading. The total body force is this value times the absolute value of the vector defining the direction of body force. In the case of centrifugal loading, the angular velocity (radians/time) should be input.

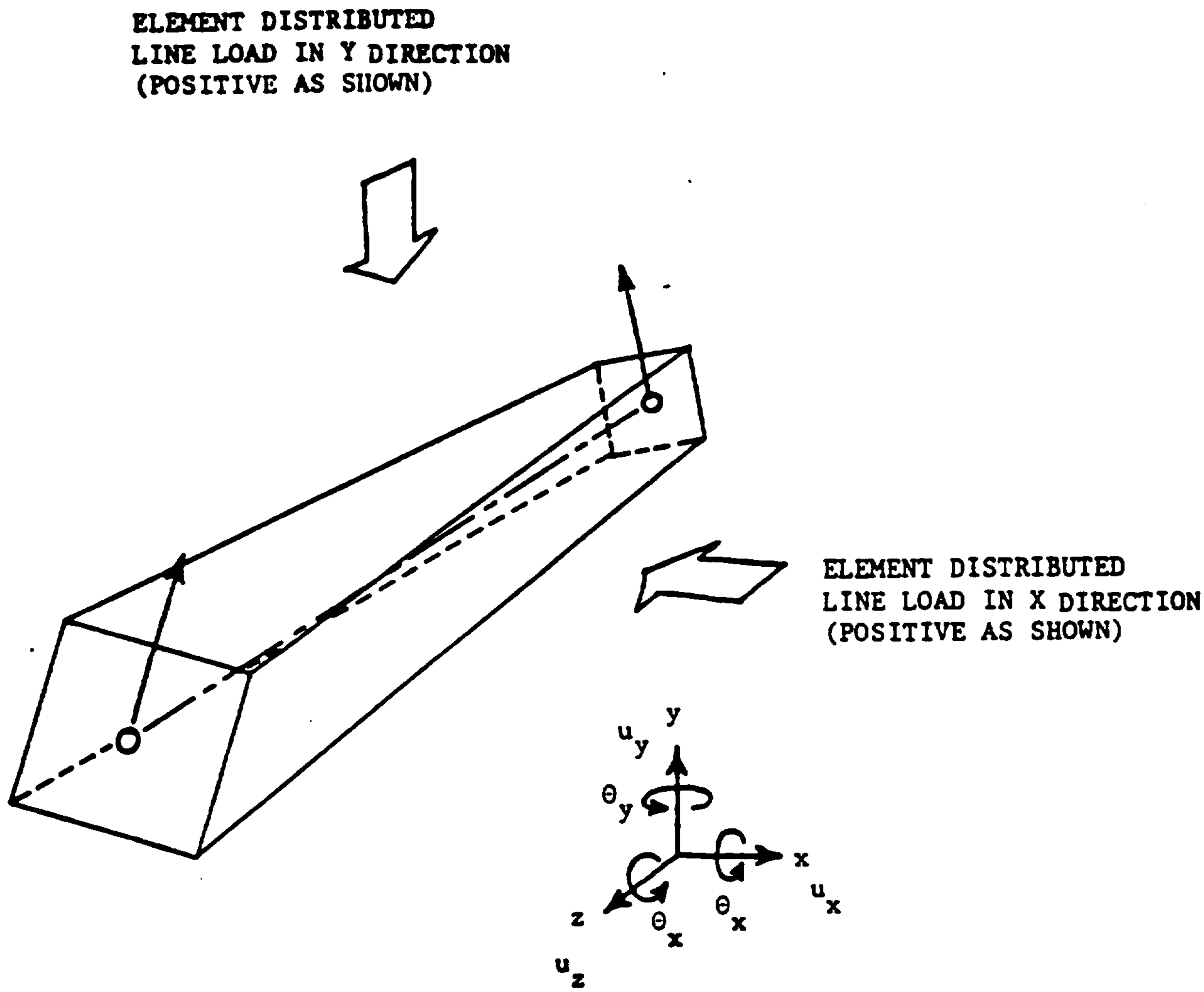

FIGURE G.6 Beam Element Traction Loading Definitions

FOUR-NODE ASSUMED STRAIN PLANE STRESS ELEMENT - MHOST ELEMENT TYPE 151

I. SUMMARY

This element is a fully integrated four-node isoparametric quadrilateral, based on an assumed strain field formulation. The element strains are computed in terms of a strain expansion of the form

$$\varepsilon_{\mathbf{xx}} = A_1 + A_2\mathbf{y} - \frac{\nu}{1-} B_2\mathbf{x}$$

$$\varepsilon_{\mathbf{yy}} = B_1 + B_2\mathbf{x} - \frac{\nu}{1-\nu} A_2\mathbf{y}$$

$$\varepsilon_{\mathbf{xy}} = C_1$$

where ($\mathbf{x}$, $\mathbf{y}$) is a local element cartesian coordinate system obtained by polar decomposition of the isoparametric mapping at the centroid of the element. This set assumed strain modes can represent all constant strain modes exactly, plus some pure bending solutions. The five generalized strain parameters $\{A_1, A_2, B_1, B_2, C_1\}$ are related to the nodal displacement vector through a weak variational form.

This element has two degrees-of-freedom per node, namely

$$\underline{u} = [u_x, u_y]^T$$

where (x, y) refer to the global cartesian coordinate system. Three strain components are defined at the nodes and the element integration points, namely

$$\underline{\varepsilon} = [\varepsilon_x, \varepsilon_y, \gamma_{xy}]^T$$

all defined in terms of the global cartesian coordinate system. The three stress components are also defined in terms of the global system, and are reported as

$$\underline{\sigma} = [\sigma_x, \sigma_y, \tau_{xy}]^T$$

at each node and/or integration point. The definition of the stress components follows the standard engineering convention.

REAL 5: Mass density.

REAL 6: Shear modulus (defaults to the isotropic value).

*DISTRIBUTEDLOAD Followed by one parameter identifying the element type (151). Subsequent data lines contain three integers and two reals:

Integer 1: The first of a series of elements which are subjected to distributed loading.

Integer 2: The last element in the series.

Integer 3: Surface traction type. Enter a value of one (1) if traction is specified on edge 1, 2 of the element.

REAL 1: Intensity of normal surface traction force on edge 1, 2 of the element. Defined as positive inward (Figure G.7) and specified in units of force per unit length.

REAL 2: Intensity of body force loading. For gravity loading, the total body force is computed by multiplying this value by the length of the vector defining the gravity direction. In the case of centrifugal loading, the angular velocity (in radians per unit time) should be specified here.

*PRESSURE Subsequent data lines contain two integers and one real:

Integer 1: The first of a series of nodes to which nodal surface pressures are applied.

Integer 2: The last node in the series.

REAL 1: Intensity of pressure loading, defined as positive into the body and specified as a force per unit length.

II. DATA PREPARATION

Specific input options which are pertinent to this element are summarized below:

1. PARAMETER DATA

*ELEMENT	Followed by an integer parameter specifying an upper bound for the number of elements. The element type (151) is specified as an integer parameter in the following data line.
*LOUBIGNAC	The third parameter, used to specify the integration rule for stiffness assembly, is ignored for this element. The element can be collapsed to form a triangle, provided that full integration is used for strain recovery and row-sum lumping used for computing the strain projection coefficients. The default values for this option are strongly recommended for all other applications.
*DISTRIBUTEDLOAD	Used without a parameter to flag the presence of element distributed loads or body forces in the model.
*PRESSURE	Used without a parameter to flag the presence of surface pressure loads specified on a nodal basis.

2. MODEL DATA

*COORDINATES	Subsequent data lines contain one integer identifying the node number, followed by two reals specifying the x and y coordinates for the node in the global cartesian coordinate system.
*PROPERTIES	Followed by one parameter identifying the element type (151). Subsequent data lines contain two integers and up to six reals:

Integer 1:	The first of a series of nodes having the same set of properties.
Integer 2:	The last node in the series.
REAL 1:	A dummy parameter.
REAL 2:	Young's modulus.
REAL 3:	Poisson's ratio.
REAL 4:	Thermal expansion coefficient.

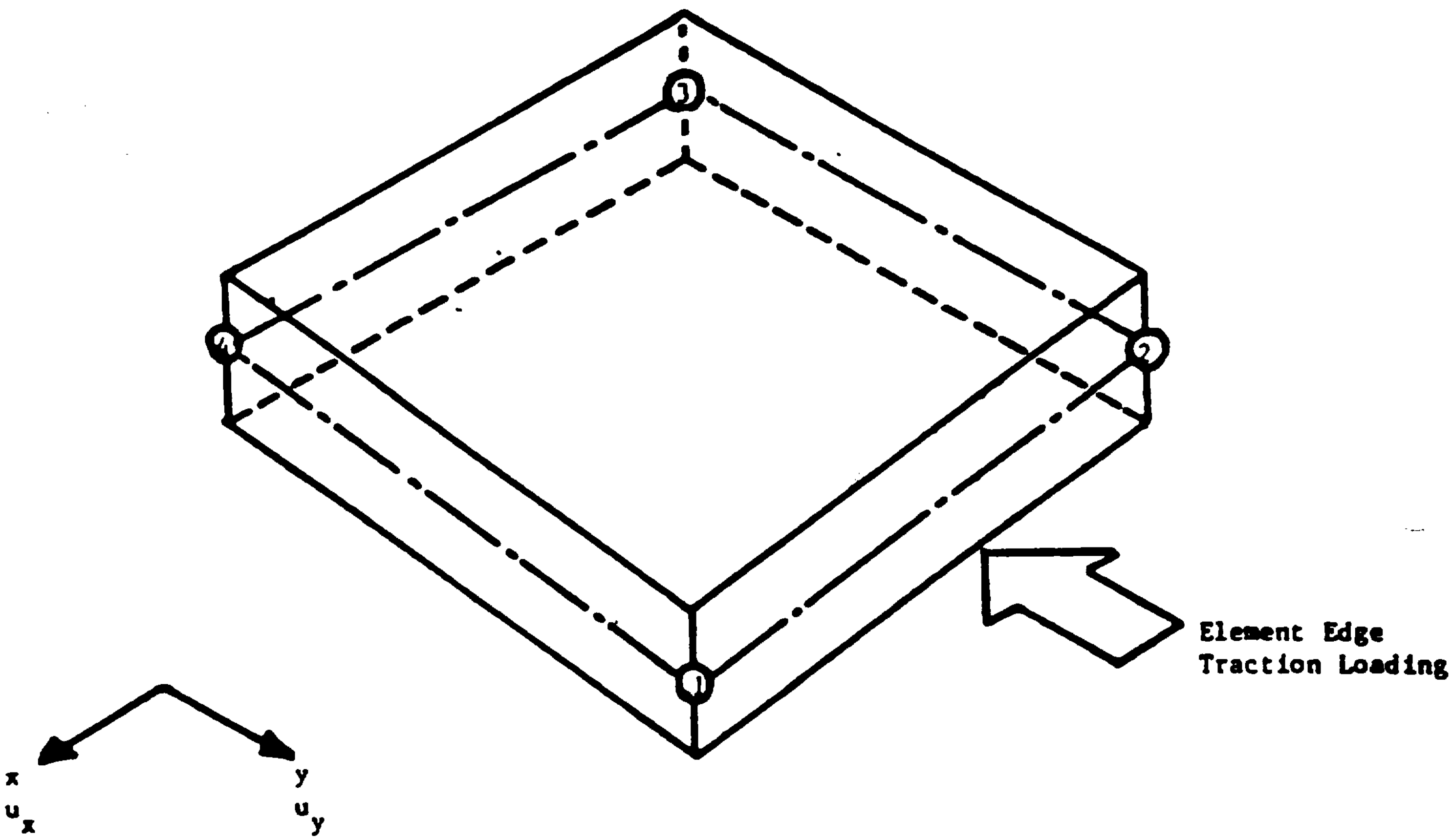

FIGURE G.7 Connectivity and Traction Loading for Element Type 151

FOUR-NODE ASSUMED STRAIN PLANE STRAIN ELEMENT - MHOST ELEMENT TYPE 152

I. SUMMARY

This element is a fully integrated four-node isoparametric quadrilateral, based on an assumed strain field formulation. The element strains are computed in terms of a strain expansion of the form

$$\varepsilon_{xx} = A_1 + A_2 y - \nu B_2 x$$

$$\varepsilon_{yy} = B_1 + B_2 x - \nu A_2 y$$

$$\varepsilon_{xy} = C_1$$

where (x, y) is a local element cartesian coordinate system obtained by polar decomposition of the isoparametric mapping at the centroid of the element. This set of assumed strain modes can represent all constant strain modes exactly, plus some pure bending solutions. The five generalized strain parameters $\{A_1, A_2, B_1, B_2, C_1\}$ are related to the nodal displacement vector through a weak variational form.

This element has two degrees-of-freedom per node, namely

$$u = [u_x, u_y]$$

where (x, y) refer to the global cartesian coordinate system. Four strain components are defined at the nodes and the element integration points, namely

$$\underline{\varepsilon} = [\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}]^T$$

all defined in terms of the global cartesian coordinate system. The four stress components are also defined in terms of the global system, and are reported as

$$\underline{\sigma} = [\sigma_x, \sigma_y, \sigma_z, \tau_{xy}]^T$$

at each node and/or integration point. The definition of the stress components follows the standard engineering convention.

II. DATA PREPARATION

Specific input options which are pertinent to this element are summarized below:

1. PARAMETER DATA

*ELEMENT — Followed by an integer parameter specifying an upper bound for the number of elements. The element type (152) is specified as an integer parameter in the following data line.

*LOUBIGNAC — The third parameter, used to specify the integration rule for stiffness assembly, is ignored for this element. The element can be collapsed to form a triangle, provided that full integration is used for strain recovery and row-sum lumping used for computing the strain projection coefficients. The default values for this option are strongly recommended for all other applications.

*DISTRIBUTEDLOAD — Used without a parameter to flag the presence of element distributed loads or body forces in the model.

*PRESSURE — Used without a parameter to flag the presence of surface pressure loads specified on a nodal basis.

2. MODEL DATA

*COORDINATES — Subsequent data lines contain one integer identifying the node number, followed by two reals specifying the x and y coordinates for the node in the global cartesian coordinate system.

*PROPERTIES — Followed by one parameter identifying the element type (152). Subsequent data lines contain two integers and up to six reals:

Integer 1: The first of a series of nodes having the same set of properties.

Integer 2: The last node in the series.

REAL 1: A dummy parameter.

REAL 2: Young's modulus.

REAL 3: Poisson's ratio.

REAL 4: Thermal expansion coefficient.

REAL 5: Mass density.

REAL 6: Shear modulus (defaults to the isotropic value).

*DISTRIBUTEDLOAD Followed by one parameter identifying the element type (152). Subsequent data lines contain three integers and two reals:

Integer 1: The first of a series of elements which are subjected to distributed loading.

Integer 2: The last element in the series.

Integer 3: Surface traction type. Enter a value of one (1) if traction is specified on edge 1, 2 of the element.

REAL 1: Intensity of normal surface traction force on edge 1, 2 of the element. Defined as positive inward (Figure G.8) and specified in units of force per unit length.

REAL 2: Intensity of body force loading. For gravity loading, the total body force is computed by multiplying this value by the length of the vector defining the gravity direction. In the case of centrifugal loading, the angular velocity (in radians per unit time) should be specified here.

*PRESSURE Subsequent data lines contain two integers and one real:

Integer 1: The first of a series of nodes to which nodal surface pressures are applied.

Integer 2: The last node in the series.

REAL 1: Intensity of pressure loading, defined as positive into the body and specified as a force per unit length.

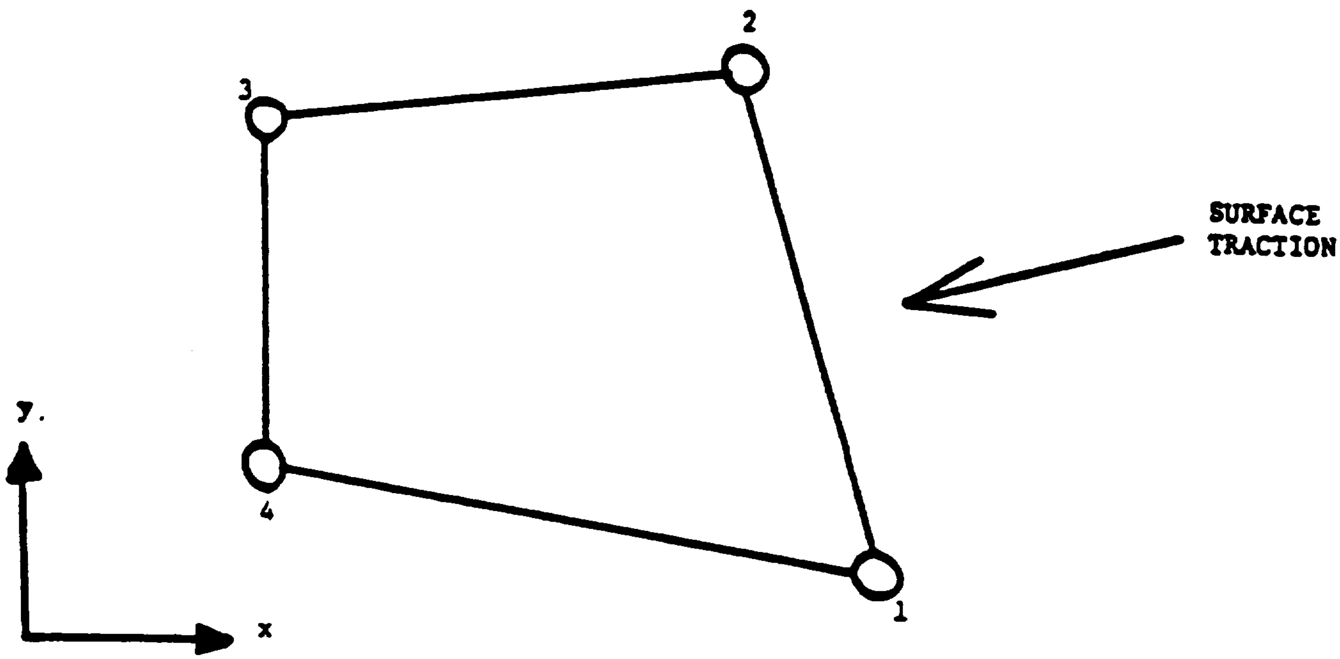

FIGURE G.8 Connectivity and Loading for Element Type 152

FOUR-NODE ASSUMED STRAIN AXISYMMETRIC ELEMENT - MHOST ELEMENT TYPE 153

I. SUMMARY

This element is a fully integrated four-node isoparametric quadrilateral, based on an assumed strain field formulation. The element strains are computed in terms of a strain expansion of the form

$$\varepsilon_{xx} = A_1 + A_2 y - \nu (B_1 + C_1) - \nu B_2 x - \nu C_2 z$$

$$\varepsilon_{yy} = B_1 + B_2 x - \nu (A_1 + C_1) - \nu A_2 y - \nu C_2 z$$

$$\varepsilon_{\theta\theta} = C_1 + C_2 z - \nu (A_1 + B_1) - \nu A_2 y - \nu B_2 x$$

$$\gamma_{xy} = D_1$$

where (x, y) is a local element orthogonal coordinate system obtained by polar decomposition of the isoparametric mapping at the centroid of the element, and z is the global symmetry axis. This set of assumed strain modes can represent all constant strain modes exactly, plus some pure bending solutions. The seven generalized strain parameters $\{A_1, A_2, B_1, B_2, C_1, C_2, D_1\}$ are related to the nodal displacement vector through a weak variational form.

This element has two degrees-of-freedom per node, namely

$$\underline{u} = [u_z, u_r]^T$$

where (z, r) refer to the global cylindrical coordinate system. Four strain components are defined at the nodes and the element integration points, namely

$$\underline{\varepsilon} = [\varepsilon_z, \varepsilon_r, \varepsilon_\theta, \gamma_{zr}]^T$$

all defined in terms of the global cylindrical coordinate system. The four stress components are also defined in terms of the global system, and are reported as

$$\underline{\sigma} = [\sigma_z, \sigma_r, \sigma_\theta, \tau_{zr}]^T$$

at each node and/or integration point. The definition of the stress components follows the standard engineering convention.

II. DATA PREPARATION

Specific input options which are pertinent to this element are summarized below:

1. PARAMETER DATA

*ELEMENT	Followed by an integer parameter specifying an upper bound for the number of elements. The element type (153) is specified as an integer parameter in the following data line.
*LOUBIGNAC	The third parameter, used to specify the integration rule for stiffness assembly, is ignored for this element. The element can be collapsed to form a triangle, provided that full integration is used for strain recovery and row-sum lumping used for computing the strain projection coefficients. The default values for this option are strongly recommended for all other applications.
*DISTRIBUTEDLOAD	Used without a parameter to flag the presence of element distributed loads or body forces in the model.
*PRESSURE	Used without a parameter to flag the presence of surface pressure loads specified on a nodal basis.

2. MODEL DATA

*COORDINATES	Subsequent data lines contain one integer identifying the node number, followed by two reals specifying the z and r coordinates for the node in the global cylindrical coordinate system.
*PROPERTIES	Followed by one parameter identifying the element type (153). Subsequent data lines contain two integers and up to five reals:

Integer 1:	The first of a series of nodes having the same set of properties.
Integer 2:	The last node in the series.
REAL 1:	A dummy parameter.
REAL 2:	Young's modulus.
REAL 3:	Poisson's ratio.
REAL 4:	Thermal expansion coefficient.

REAL 5: Mass density.

*DISTRIBUTEDLOAD Followed by one parameter identifying the element type (153). Subsequent data lines contain three integers and two reals:

Integer 1: The first of a series of elements which are subjected to distributed loading.

Integer 2: The last element in the series.

Integer 3: Surface traction type. Enter a value of one (1) if traction is specified on edge 1, 2 of the element.

REAL 1: Intensity of normal surface traction force on edge 1, 2 of the element. Defined as positive inward (Figure G.9) and specified in units of force per unit length.

REAL 2: Intensity of body force loading. For gravity loading, the total body force is computed by multiplying this value by the length of the vector defining the gravity direction. In the case of centrifugal loading, the angular velocity (in radians per unit time) should be specified here. The gravity force can only be applied in the axial direction (along the global z-axis), and the centrifugal force about the same axis.

*PRESSURE Subsequent data lines contain two integers and one real:

Integer 1: The first of a series of nodes to which nodal surface pressures are applied.

Integer 2: The last node in the series.

REAL 1: Intensity of pressure loading, defined as positive into the body and specified as a force per unit area.

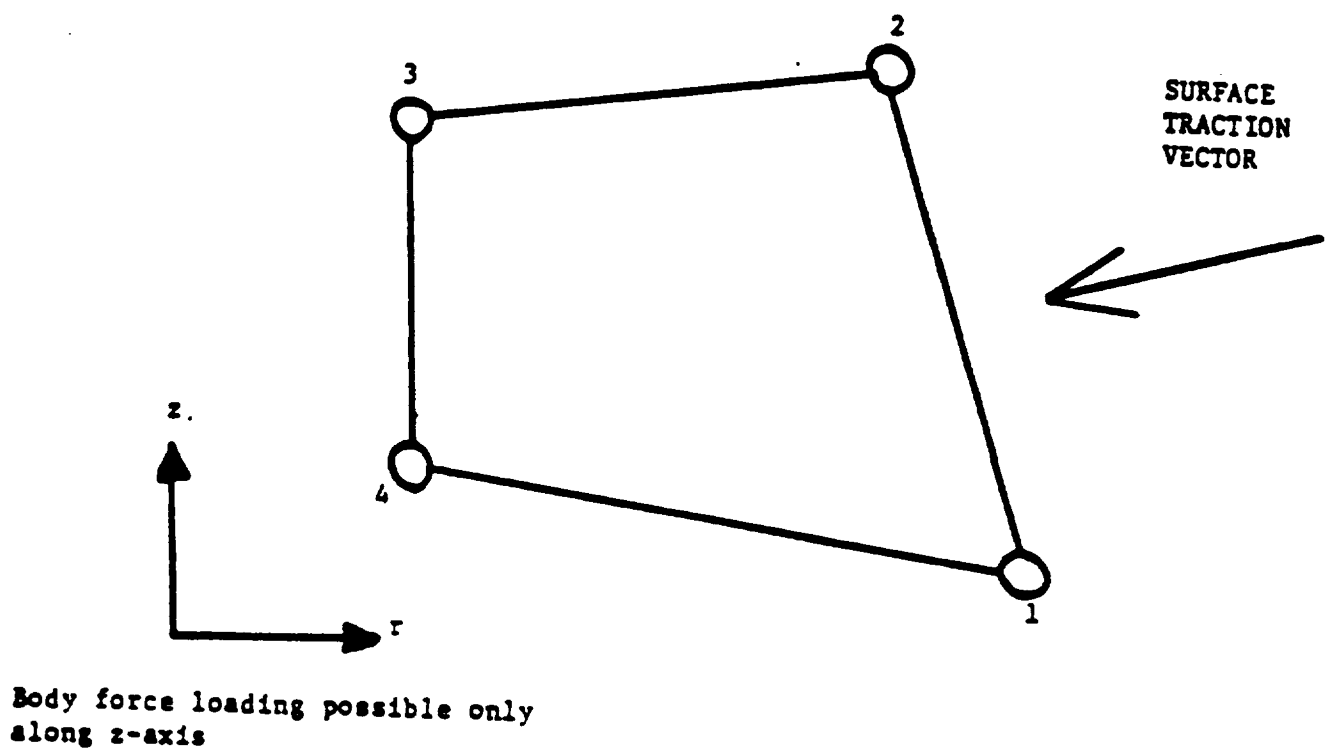

FIGURE G.9 Connectivity and Loading for Element Type 153

EIGHT-NODE ASSUMED STRAIN SOLID ELEMENT - MHOST ELEMENT TYPE 154

I. SUMMARY

This element is a fully integrated eight-node isoparametric solid, based on an assumed strain field formulation. The element strains are computed in terms of a strain expansion of the form

$$\varepsilon_{xx} = A_1 + A_2 y + A_3 z + A_4 yz + \frac{\nu}{1-\nu}(B_3 + C_2)\, x - \nu B_4 zx - \nu C_4 xy$$

$$\varepsilon_{yy} = B_1 + B_2 z + B_3 x + B_4 zx + \frac{\nu}{1-\nu}(C_3 + A_2)\, y - \nu C_4 xy - \nu A_4 yz$$

$$\varepsilon_{zz} = C_1 + C_2 x + C_3 y + C_4 xy + \frac{\nu}{1-\nu}(A_3 + B_2)\, z - \nu A_4 yz - \nu B_4 zx$$

$$\varepsilon_{xy} = D_1 + D_2 z \qquad \varepsilon_{yz} = E_1 + E_2 x \qquad \varepsilon_{zx} = F_1 + F_2 y$$

where (x, y, z) is a local element cartesian coordinate system obtained by polar decomposition of the isoparametric mapping at the centroid of the element. This set assumed strain modes can represent all constant strain modes exactly, plus some pure bending solutions. The fifteen generalized strain parameters $\{A_1, A_2, \ldots, F_2\}$ are related to the nodal displacement vector through a weak variational form.

This element has three degrees-of-freedom per node, namely

$$\underline{u} = [u_x, u_y, u_z]^T$$

where (x, y, z) refer to the global cartesian coordinate system. Six strain components are defined at the nodes and the element integration points, namely

$$\underline{\varepsilon} = [\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{zx}]^T$$

all defined in terms of the global cartesian coordinate system. The six stress components are also defined in terms of the global system, and are reported as

$$\underline{\sigma} = [\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx}]^T$$

at each node and/or integration point. The definition of the stress components follows the standard engineering convention.

II. DATA PREPARATION

Specific input options which are pertinent to this element are summarized below:

1. PARAMETER DATA

*ELEMENT — Followed by an integer parameter specifying an upper bound for the number of elements. The element type (154) is specified as an integer parameter in the following data line.

*LOUBIGNAC — The third parameter, used to specify the integration rule for stiffness assembly, is ignored for this element. The element can be collapsed to form a wedge or tetrahedron provided that full integration is used for strain recovery and row-sum lumping used for computing the strain projection coefficients. The default values for this option are strongly recommended for all other applications.

*DISTRIBUTEDLOAD — Used without a parameter to flag the presence of element distributed loads or body forces in the model.

*PRESSURE — Used without a parameter to flag the presence of surface pressure loads specified on a nodal basis.

2. MODEL DATA

*COORDINATES — Subsequent data lines contain one integer identifying the node number, followed by three reals specifying the x, y and z coordinates for the node in the global cartesian coordinate system.

*PROPERTIES — Followed by one parameter identifying the element type (154). Subsequent data lines contain two integers and up to six reals:

Integer 1: The first of a series of nodes having the same set of properties.

Integer 2: The last node in the series.

REAL 1: A dummy parameter.

REAL 2: Young's modulus.

REAL 3: Poisson's ratio.

REAL 4: Thermal expansion coefficient.

REAL 5: Mass density.

REAL 6: Shear modulus (defaults to the isotropic value).

*DISTRIBUTEDLOAD Followed by one parameter identifying the element type (154). Subsequent data lines contain three integers and two reals:

Integer 1: The first of a series of elements which are subjected to distributed loading.

Integer 2: The last element in the series.

Integer 3: Surface traction type. Enter a value of one (1) if traction is specified on the 1, 2, 3, 4 element face.

REAL 1: Intensity of normal surface traction force on face 1, 2, 3, 4 of the element. Defined as positive inward (Figure G.10) and specified in units of force per unit area.

REAL 2: Intensity of body force loading. For gravity loading, the total body force is computed by multiplying this value by the length of the vector defining the gravity direction. In the case of centrifugal loading, the angular velocity (in radians per unit time) should be specified here.

*PRESSURE Subsequent data lines contain two integers and one real:

Integer 1: The first of a series of nodes to which nodal surface pressures are applied.

Integer 2: The last node in the series.

REAL 1: Intensity of pressure loading, defined as positive into the body and specified as a force per unit area.

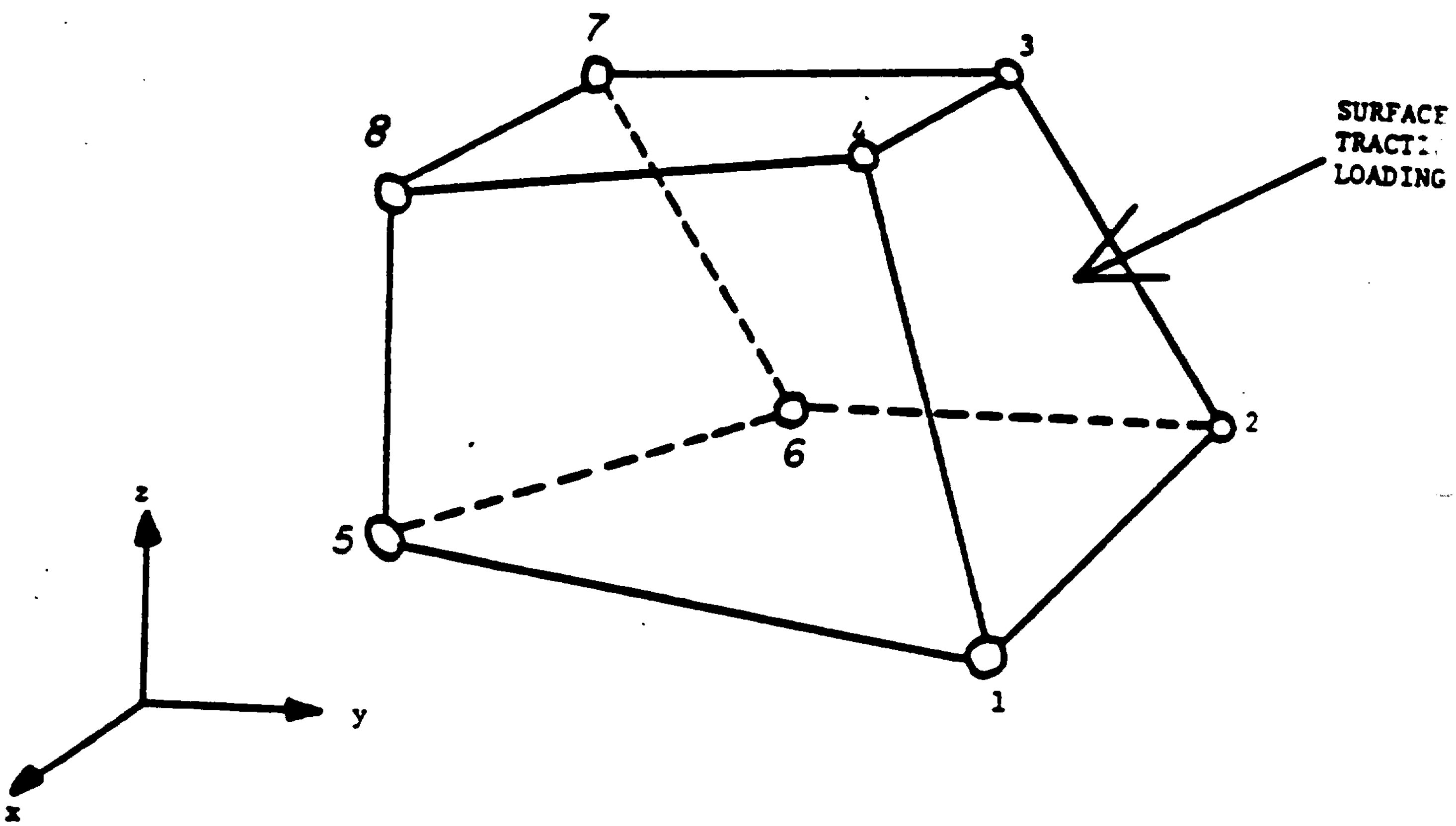

FIGURE G.10 Connectivity and Loading for Element Type 154

H. LINE PRINTER OUTPUT

The following information is produced on the line printer file.

1. MHOST logo, date and version number. On certain systems the date and time when the data is processed is also reported.

2. Echo prints of data lines in parameter data section, and the core storage used by the data input processors.

3. Echo prints of data lines in model data section, unless *NOECHO option is turned on.

4. Summary of the parameter data and model data interpretation.

5. The record of execution for the zeroth increment including the size of system matrix, the core storage used by the finite element solution processor followed by the condition of stiffness matrix and convergence of iterative procedure.

   As a measure of convergence, the program reports:

   a) Relative errors in residual.
   b) Absolution errors in residual.
   c) Root mean square error in displacement.
   d) Root mean square error in energy (which is the element-weighted-averaged relative error in residual).

6. When convergence or the maximum number of iterations is reached, the latest result has been printed in this file.

   The quantities specified by the PRINTOPTION data in the model data section are printed accordingly.

   a) For a vectorial quantities, its physical components are printed with respect to the global coordinate system except for points at which a coordinate transformation has been explicitly defined. At the node where the user has defined a coordinate transformation, all vector components are printed in the user defined coordinate system.

   b) For tensorial quantities, the von Mises intensity (the octahedral value or the second invariant) is printed out as the zeroth component.

7. When the execution fails, the program stops after printing the following error logo. In most cases, an explanatory error message is provided by MHOST.

```
CCCCCCCCCC  CCCCCCCC    CCCCCCCC     CCCCCCCC   CCCCCCCC
CCCCCCCCCC  CCCCCCCCC   CCCCCCCCC   CCCCCCCCCC  CCCCCCCCC
CC          CC      CC  CC      CC  CC      CC  CC      CC
CC          CC      CC  CC      CC  CC      CC  CC      CC
CCCCCCCC    CCCCCCCCC   CCCCCCCCC   CC      CC  CCCCCCCCC
CCCCCCCC    CCCCCCCC    CCCCCCCC    CC      CC  CCCCCCCC
CC          CC    CC    CC    CC    CC      CC  CC    CC
CC          CC     CC   CC     CC   CC      CC  CC     CC
CCCCCCCCCC  CC      CC  CC      CC  CCCCCCCCCC  CC      CC
CCCCCCCCCC  CC      CC  CC      CC   CCCCCCCC   CC      CC
```

8. If a modal analysis is performed, the following information will be printed for each eigenvalue/eigenvector pair:

   a) The eigenvalue number and value.

   b) The corresponding frequency in both radians per time and cycles per time.

   c) The corresponding eigenvector, normalized so that the L norm (square root of the sum of the squares of all elements) is 1.0.

   d) The generalized mass associated with this normalization of the eigenvector.

9) When the subelement refinement option has been requested, the following information is generated and reported:

   a) At the end of the bulk data print out section the subelement mesh data is reported for each global element, including the physical and isoparametric nodal coordinates and element connectivities.

   b) After the global solution is reported, the same quantities are obtained in the subelement region and printed for each global element.

I. LOG FILE

The progress of execution is recorded in the log file (FORTRAN Unit No. 1) and printed st the terminal when the job is executed interactively on PRIME (Primos), IBM (VM/CMS), and VAX (VMS) operating systems. An example is found in Section A.1 of Volume II.

For unsuccessful executions, a brief error summary is provided in the log file. Detailed error diagnostics are only provided in the line printer file.

J. POST PROCESSING FILE

When *POST is entered in the parameter data block, MHOST results at the end of each specified increment are written on FORTRAN Unit No. 19 in formatted (80 bytes/record) form. No user control of the contents of this file is possible.

All the information is reported at nodal points. The header records and the format specification for the mesh data are compatible with the MARC General Purpose Finite Element Program (Version K2) formatted post tape. The full specifications of this file can be found in MHOST Version 4.2 Systems' Manual.

The incremental solution data is written as both element data and nodal data. The element data records consist of the stress and various strain components along with their invariants at nodes. This data is in 6G13.6 format. The nodal displacement and reaction force vector components are written in the same format as in MARC K1. MENTAT versions number 5.0 and higher are equipped to read the MHOST Version 4.2 post tape.

For details of the element data specification see MHOST Systems' Manual.

K. MHOST INDEX

KEYWORD		PAGE
	- PARAMETER DATA -	
C.1	*ELEMENTS	17
C.2	*NODES	18
C.3	*BOUNDARY	19
C.4	*TYING	19
C.5	*TRANSFORMATIONS	19
C.6	*FORCES	19-20
C.7	*PRINTSETS	20
C.8	*RESTART	20
C.9	*TEMPERATURE	20
C.10	*LOUBIGNAC	20-21
C.11	*STRESS	21
C.12	*DYNAMIC	21
C.13	*OPTIMIZE	21-22
C.14	*CREEP	22
C.15	*ANISOTROPY	22
C.16	*MODAL	22
C.17	*BUCKLE	22-23
C.18	*THERMAL	23
C.19	*CONSTITUTIVE	23
C.20	*DISTRIBUTEDLOAD	23
C.21	*REPORT	23-24
C.22	*TANGENT	24
C.23	*UTHERM	24
C.24	*UFORCE	24
C.25	*UTEMP	24
C.26	*UCOEF	24
C.27	*UPRESS	24
C.28	*UHOOK	25
C.29	*UDERIV	25
C.30	*SCHEME	25
C.31	*PERIODICLOADING	25
C.32	*DUPLICATEMODE	25
C.33	*POST	25-26
C.34	*UBOUN	26
C.35	*BANDMATRIX	26
C.36	*FRONTALSOLUTION	26
C.37	*DEFORMATION	26-27
C.38	*EMBED	27
C.39	*WKSLP	27
C.40	*BEAMSECTION	27
C.41	*DISPLACEMENTMETHOD	27-28
C.42	*SHIFT	28
C.43	*BFGS	28
C.44	*SPRINGS	28
C.45	*DASHPOTS	28-29
C.46	*MASSES	29
C.47	*LINESEARCH	29
C.48	*SECANTNEWTON	29
C.49	*HARMONICLOAD	29-30
C.50	*BASEEXCITATION	30
C.51	*COMPOSITE	30
C.52	*STIFFENING	30
C.53	*CENTRIFUGALMASS	30-31
C.54	*CONJUGATEGRADIENT	31
C.55	*NOECHO	31
C.56	*POWERSPECTRUM	31-32
C.57	*PULSELOAD	32
C.58	*PRESSURE	32
C.59	*HARDENING_SLOPE	32-33
C.60	*LARGE_DISPLACEMENT	33
C.61	*FINITE_STRAIN	33
C.62	*FOLLOWER_FORCE	33
C.63	*END	33
	- MODEL DATA -	
D.1	*ELEMENTS	36
D.2	*BOUNDARY	36
D.3	*TYING	36-37
D.4	*FORCES	37-38
D.5	*TRANSFORMATIONS	38
D.6	*ITERATIONS	38-39
D.7	*PROPERTIES	39-40
D.8	*COORDINATES	40
D.10	*INCREMENTS	40-41
D.11	*TEMPERATURES	41
D.12	*STRESS	41
D.13	*PRINTOPTION	41-42
D.14	*SAVE	43
D.15	*DEBUG	43
D.16	*END	43
D.17	*STOP	43
D.18	*WORKHARD	43
D.19	*DISTRIBUTEDLOAD	43-44
D.20	*UXFORD	44
D.21	*BODYFORCE	44-45
D.22	*DUPLICATENODE	45
D.23	*TIME	45
D.24	*DAMPING	45-46
D.26	*BEAMSECTION	46
D.27	*SHIFT	46-47
D.28	*SPRINGS	47
D.29	*DASHPOTS	47-48
D.30	*MASSES	48
D.31	*HARMONICLOAD	48
D.32	*BASEXCITATION	49
D.33	*LAMINATES	49-50
D.34	*DMATRIX	50

KEYWORD		PAGE
D.35	*ORIENTATION	50-51
D.36	*HOLE	52
D.37	*POWER	52
D.38	*SPECTRUM	52
D.39	*BREAKPOINTS	52-53
D.40	*ENDPOWER	53
D.41	*PULSELOAD	53
D.42	*TIMEHISTORY	53
D.43	*NODALDEFINITION	53-54
D.44	*ENDPULSELOAD	54
D.45	*INCLUDE	54
D.46	*RESUME	54
D.49	*PRESSURE	54-55
D.50	FOLLOWER_FORCE	55
	- INCREMENTAL DATA -	
E.1	*BOUNDARY	57
E.2	*TYING	57
E.3	*FORCES	57
E.4	*DISTRIBUTEDLOAD	57
E.5	*TEMPERATURES	57
E.6	*STRESS	57
E.7	*ITERATIONS	57-58
E.8	*PROPORTIONAL	58
E.9	*AUTOINCREMENT	58
E.10	*SAVE	58
E.11	*DEBUG	58
E.12	*PRINTOPTION	58-59
E.13	*TIME	59
E.14	*STOP	59
E.15	*END	59
E.16	*BODYFORCE	59
E.17	*INITIALCONDITION	59
E.18	*DISPLACEMENT	59
E.19	*VELOCITY	59-60
E.20	*ACCELERATION	60
E.21	*PERIODICLOADING	60
E.22	*ENDINITIALCONDITION	60
E.23	*FOLLOWER_FORCE	61