USER'S MANUAL FOR

F : FINITE
E : ELEMENT
N : NONLINEAR (& LINEAR)
D : DYNAMIC (& STATIC)
A : ANALYSIS
C : CODE

Written By: Colby C. Swan, Assoc. Prof.
Department of Civil & Environmental Engineering
The University of Iowa
Iowa City, IA 52242-1527

Last Update: September 22, 2002
Language: FORTRAN 77

DESCRIPTION OF THE COMPUTER CODE FENDAC

Colby C. Swan
Dept. of Civil & Env. Engineering
The University of Iowa
Iowa City, IA 52242-1527
(319) 335-5831

FENDAC is a finite element program for the static and transient response of linear and nonlinear two- and three-dimensional systems. In particular, it offers transient capabilities for both hyperbolic and parabolic initial value problems in structural, continuum, porous medium applications, and quasi-static capabilities for many types of elliptic boundary value problems. There are no restrictions on the size of problems that can be solved with the program, other than those presented by machine limitations. While maintaining a large system capacity, small problems can also be efficiently handled with this code. In both static and transient analyses, implicit-explicit predictor-multicorrector schemes are employed. The solution algorithms currently employed to solve the nodal force balance equations include: Newton-Raphson, modified Newton, Quasi-Newton (BFGS), memoriless Quasi-Newton, and conjugate gradient methods. Each method can be used with or without line-searching. Additional features which are available in FENDAC are:

- Highly vectorized sparse, skyline and profile Choleski and/or Crout equation solvers
- Symmetric and non-symmetric skyline Crout equation solvers
- Options for prescribing essential boundary conditions as displacements, velocities, or accelerations
- Isoparametric data generation schemes for nodes, elements and other input data
- Node number optimization to minimize the profile of the global tangent stiffness matrix
- Nodal enslavement capabilities
- Periodic boundary conditions and volume-averaging for homogenization of heterogeneous media
- Topology optimization capabilities for optimum design of both structures and composite materials.
- Complete restarting capabilities
- Local error estimation for usage in adaptive mesh refinement.
- Nodal and element time histories
- Eigenvalue and Eigenmode Analysis.
- Complex frequency domain analysis of damped systems.
- Poro-elastic analysis in two and three dimensions.
- Interactive color mesh post-processing capabilities for both 2D and 3D analysis (Currently available only in SGI Environments)

Table of Contents

Page

1. TITLE LINE

Each data file should begin with a title line describing the problem being analyzed. If longer than 80 characters, the line will be truncated to 80 characters. The title line will appear in all output files, plots and graphics that are generated as a result of the computation being performed.

1.1 Formatting of Data

The initial control lines of data are optionally terminated with a blank line. Subsequent input modules, use **keywords** to mark the beginning of data for that module. Accordingly, after the initial control lines, all of the subsequent data input modules have the form:

- **KEYWORD** (1 line) with the actual word varying for each module.
- input data (multiple lines)
- one or more blank lines to terminate the input module

2. INITIAL CONTROL LINES

2.1 First Control Line

Immediately following the title line, FENDAC expects to find two control lines with no blank lines between them. The first control line has the format 20i5 and the values prescribed are as follows:

Columns	Variable	Description
1- 5	NUMNP	Number of nodes in mesh; (must be>0)
6-10	NSD	Number of spatial dimensions ($1 \leq NSD \leq 3$)
11-15	NDOF	Degrees of freedom per node ($1 \leq NDOF \leq 6$)
16-20	NUMEG	Number of element groups (≥ 1)
21-25	ISYMM	Symmetry indicator for global tangent
		•0 if symmetric, 1 otherwise
26-30	MODE	Execution mode:
		0 for data check;
		1 for normal execution;
		2 for rank check of stiffness matrix;
		3 for eigensolving mode;
		requires additional input following (KW = EIGENSOL)
		4 for complex frequency domain analysis;
31-35	NDOUT	Number of nodal time histories
		•Requires additional input following NODEHIST keyword
36-40	ISHELL	Shell analysis indicator (1 when shells included, 0 otherwise)
41-45	IDUMP	Perform nodal/element output every IDUMP steps
46-50	IAVHIST	1 for homogenization, 0 otherwise
51-55	IREST	0 for starting computation, 1 for restarting
56-60	NSLAVE	Number of lines of type-1 nodal enslavement data to read
		•Requires additional input following SLAVENOD keyword
61-65	LDVA	Mode for application of essential bc data
		•1 = displacements, 2 = velocities, 3 = accelerations
66-70	IOPTZ	Node number optimization to minimize profile
		•-1 Perform optimization (do not print map)
		• 0 Do not perform optimization
		• 1 Perform optimization (print map)

(See following page for continuation of variable descriptions . . .)

(Continued Initial Control Line Data . . .)

Columns	Variable	Description
71-75	IBOUND	If $\neq 0$, periodic boundary conditions during homogenization if < 0: Type I conditions (strain-controlled) • Requires input following PERIODIC keyword if > 0: Type II conditions (not currently used)
76-80	IDESIGN	If > 0, perform design sensitivity for topology optimization •Requires additional input following TOPOLOGY keyword
81-85	ILIMIT	If > 0, perform limit analysis computations (see [12,13]) Value is the number of times time step will be quartered when convergence is not achieved
86-90	ICORE	Code for storage of element group data If 0, (Default) Use direct access file to store multiple element group data If 1, store all element group data in core.
91-95	NSLAVE2	Number of lines of type-2 nodal enslavement data to read •Requires additional input following SLAV2NOD keyword
96-100	IREFINE	Code for adaptive mesh refinement sensitivities If 0, (Default) Do not compute sensitivities If 1, Compute the sensitivities

2.2 Second Control Line

The second control line which follows immediately after the first has the format 6i5,5f10,i5 and the values prescribed are:

Columns	Variable	Description
1- 5	ITYPE	Type of partial differential equation being solved
		•1 (elliptic) quasi-static or steady state
		•2 (parabolic) transient diffusion
		•3 (hyperbolic) transient vibrations or waves
6-10	NTS	Number of time/load steps for analysis (>0)
11-15	NLC	Number of load cases to read in following LOADCASE
16-20	NLTF	Number of load-time functions to read following LOADTIME
21-25	NLS	Max number of steps per load-time function
26-30	IMASS	•0 for consistent masses (implicit transient)
		•1 for diagonal mass matrix (explicit transient)
31-40	ALPHA	Newmark velocity integration parameter ($0 \leq \alpha \leq 1$)
41-50	BETA	Newmark acceleration integration parameter ($0 \leq \beta \leq 1$)
51-60	DT	Time step for analysis (> 0)
61-70	TIM	Time at which analysis begins (for starting computations)
71-80	TMAX	Maximum problem time
81-85	LCAST	Load time function number to modulate time step
		•if 0, time step is constant

3. ITERATION/SOLUTION ALGORITHM (KW = ITERALGO)

3.1 Keyword

This module of input follows the keyword **ITERALGO** which must be left-justified and spelled in uppercase letters on a separate line.

3.2 Specified Data

The information for the iteration solution algorithm to be employed follows on a line immediately following the left-justified keyword **ITERALGO.** The format for the data is 2i5,2f10,i5,f10,3i5 and the actual variable values prescribed are as follows:

Columns	Variable	Description
1- 5	NITER	Maximum number of iterations allowed per time/load step
6-10	NFAC	Reform and factorize the global tangent every NFAC iterations
11-20	RTOL	Absolute convergence tolerance for residual vector (inf) norm
21-30	DTOL	Absolute convergence tolerance for incremental kinematic vector norm
31-35	NLSCH	Max. number of line search iterations allowed per Newton iteration
		•if = 0, then no search is ever performed
		•if $>$ 1, a search is performed
36-45	STOL	Line search criterion tolerance
46-50	ISOLVE	Equation Solving Algorithm
		•-4 Vectorized sparse GPS Choleski (NASA)
		•-3 Vectorized sparse VSS Choleski (Solversoft)
		•-2 Vectorized skyline Choleski
		•-1 Vectorized profile Choleski
		• 0 Skyline Crout
		• 2 Memoriless BFGS (no Powell restarts)
		• 3 Memoriless BFGS with Powell restarts
		• 4 Out-of-core Crout (currently unavailable)
		• 5 Jacobi-Preconditioned Conjugate Gradient
51-55	INEWTON	Global Solution Algorithm
		•0 Newton/Modified Newton
		•1 Standard BFGS
		(Available only with ISOLVE $\leq$ 0.)
56-60	IOPTSTR	if 1, create graphics file to show structure of tangent matrix

3.3 Termination

Terminate this input module with a blank line.

4. NODAL COORDINATE DATA (KW = NODCOORD)

4.1 Keyword

The keyword for this input module is **NODCOORD.**

4.2 Nodal Coordinate Lines

Nodal coordinate data begins on the line immediately following the keyword **NODCOORD**. The first line is always a Nodal Coordinate Line having the format (2i5, NSD*(ISHELL+1)*f16).

Columns	Variable	Description
1- 5	N	Node number ($1 \leq N \leq NUMNP$)
6-10	NUMGP	Number of generation points (≥ 0)
		if 0, no generation lines follow
		if > 0, NUMGP-1 generation lines follow
11- 26	x(1,N)	x_1 coordinate of node N (x_1^+ for shell nodes)
27- 42	x(2,N)	x_2 coordinate of node N (x_2^+ for shell nodes)
43- 58	x(3,N)	x_3 coordinate of node N (x_3^+ for shell nodes)
59- 74	x(4,N)	x_1 coordinate of node N (x_1^- for shell nodes)
75- 90	x(5,N)	x_2 coordinate of node N (x_2^- for shell nodes)
91-106	x(6,N)	x_3 coordinate of node N (x_3^- for shell nodes)

Notes:

1. The coordinate of each node must be defined, but the nodes need not be listed in order.
2. If NUMGP is greater than zero, this data line initiates an isoparametric data generation sequence. Lines 2 to NUMGP following the nodal coordinate line define the coordinates of the additional generation points (Section 4.3). The final data line of the generation sequence is the nodal increment line (Section 4.4). After a generation sequence is completed, additional nodal coordinate lines or generation sequences may follow.
3. The generation may be performed along a line or curve (for NSD = 1,2,3), over a surface (for NSD = 2,3), or over a volume (for NSD = 3). A description of each of these options is provided below:

4.2.1 Generation along a line

For NUMGP=2, the line is defined by 2 generation points while the physical space may be 1, 2, or 3 dimensional (Figure 4.1). The nodal points placed along the line by the generation process will be equally spaced and their coordinates will be as follows:

$$\mathbf{x}_A = (1 - \xi_A)\mathbf{x}_1 + \xi_A\mathbf{x}_2, \quad \text{where} \quad -1 \leq \xi_A \leq 1.$$

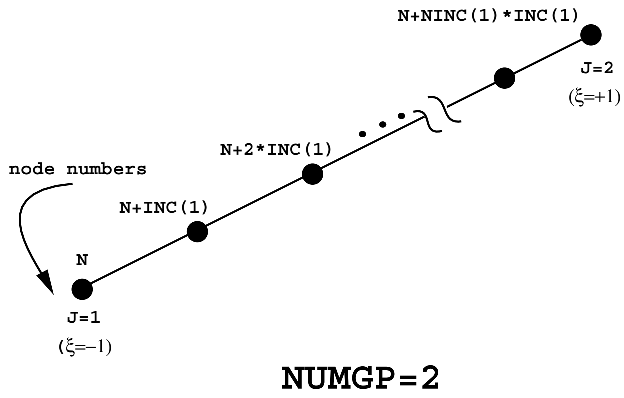


Figure 4.1: Linear generation of nodal points along a line.

4.2.2 Generation along a curve

For NUMGP=3, the curve is defined by 3 generation points while the physical space may again be 1, 2, or 3 dimensional (Figure 4.2). The first two points define the endpoints of the curve, while the third point is an interior point. Quadratic interpolation is employed, and graded nodal spacing can be achieved by placing the third generation point off center. The nodal points are placed along the curve in accordance with the relation

$$\mathbf{x}_A = \frac{1}{2}\xi_A(\xi_A - 1)\mathbf{x}_1 + \frac{1}{2}\xi_A(\xi_A + 1)\mathbf{x}_2 + (1 - \xi_A^2)\mathbf{x}_3, \quad \text{where} \quad -1 \le \xi_A \le 1.$$

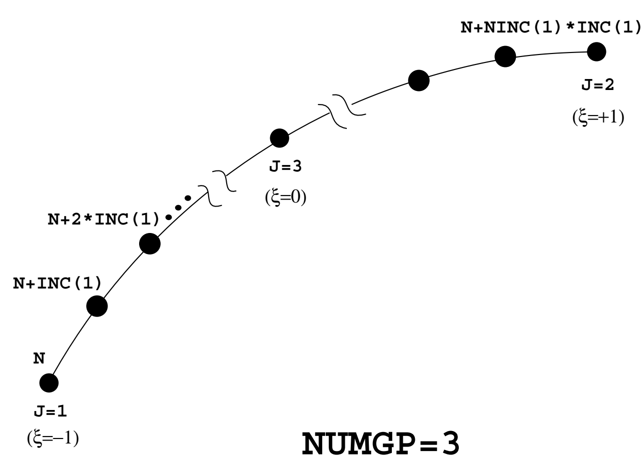


Figure 4.2: Quadratic generation of nodal points along a curve.

4.2.3 Bilinear generation over a surface

For NSD = 2 or 3, bilinear generation can be performed over a surface where the surface is defined by 4 generation points (Figure 4.3). Each nodal point placed on the generated surface is given coordinates in accordance with the relation:

$$\mathbf{x}(\xi, \eta) = \sum_{J=1}^{4} N_J(\xi, \eta)\mathbf{x}_J,$$

where

$$N_1(\xi, \eta) = \frac{1}{4}(1 - \xi)(1 - \eta)$$

$$N_2(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 - \eta)$$

$$N_3(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 + \eta)$$

$$N_4(\xi, \eta) = \frac{1}{4}(1 - \xi)(1 + \eta)$$

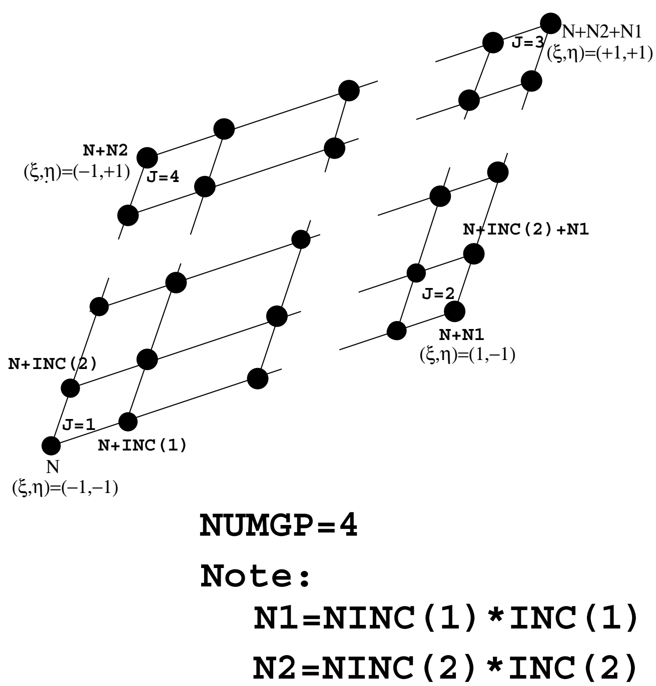


Figure 4.3: Bilinear generation of nodal points on a surface.

4.2.4 Biquadratic serendipity generation over a surface

For NSD= 2 and NUMGP =8, biquadratic generation will be performed over a surface (Figure 4.4). Generation points 1-4 define the corners of the surface region, while generation points 5-8 define interior points on the surface's four edges. Graded nodal spacing may be achieved by placing points 5–8 off center. Note that generation points 5–8 need not necessarily coincide with nodal points.

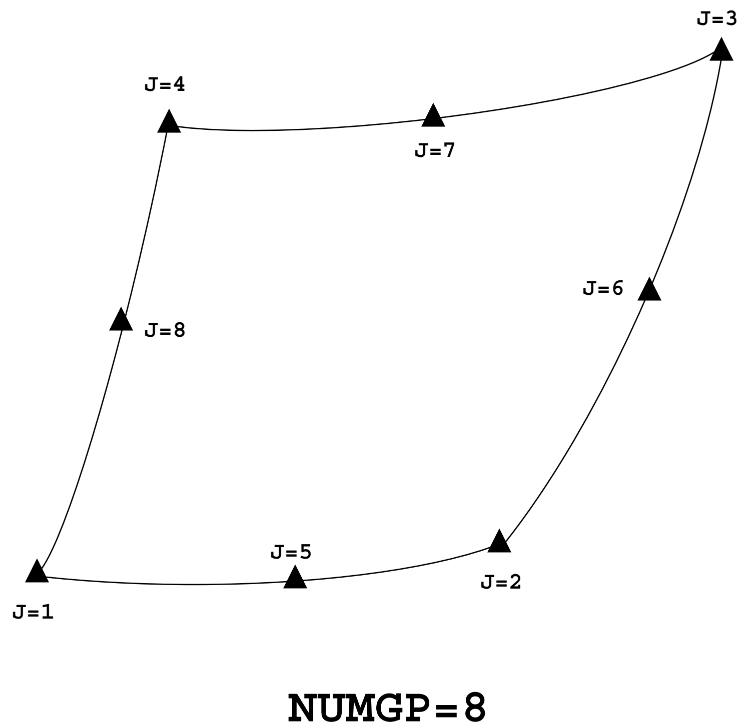


Figure 4.4: Biquadratic serendipity generation of nodal points on a surface.

4.2.5 Trilinear generation over a volume

For three-dimensional problems (NSD=3), 8 point generation (NUMGP=8) can be used to achieve trilinear placement of nodes over a brick shaped volume (Figure 4.5). This method gives equally spaced nodal along each edge of the generated domain.

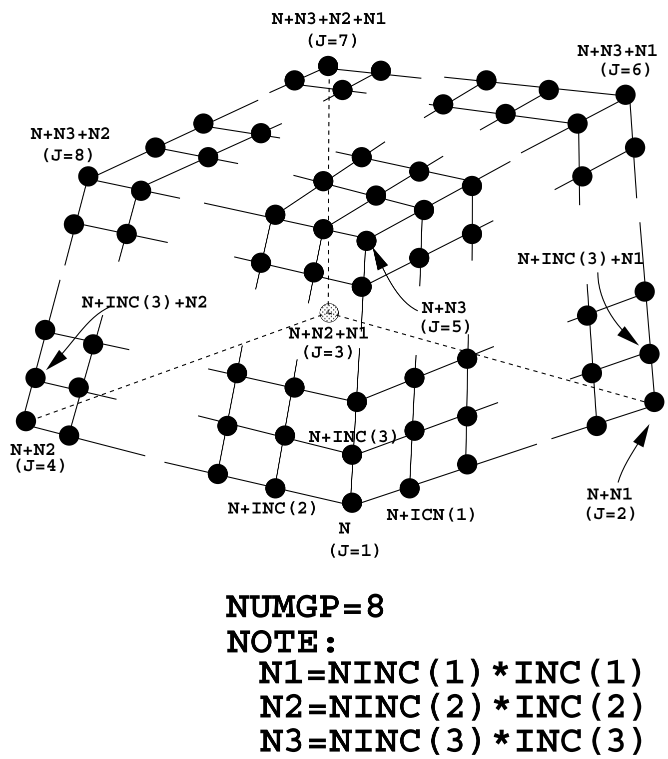


Figure 4.5: Trilinear generation of nodal points over a volume.

4.2.6 Triquadratic generation over a volume

For three-dimensional problems (NSD=3), 20 point generation (NUMGP=20) can be used to achieve triquadratic placement of nodes over a volume (Figure 4.6). This method can give curved edges and graded nodal spacings along each edge of the generated domain.

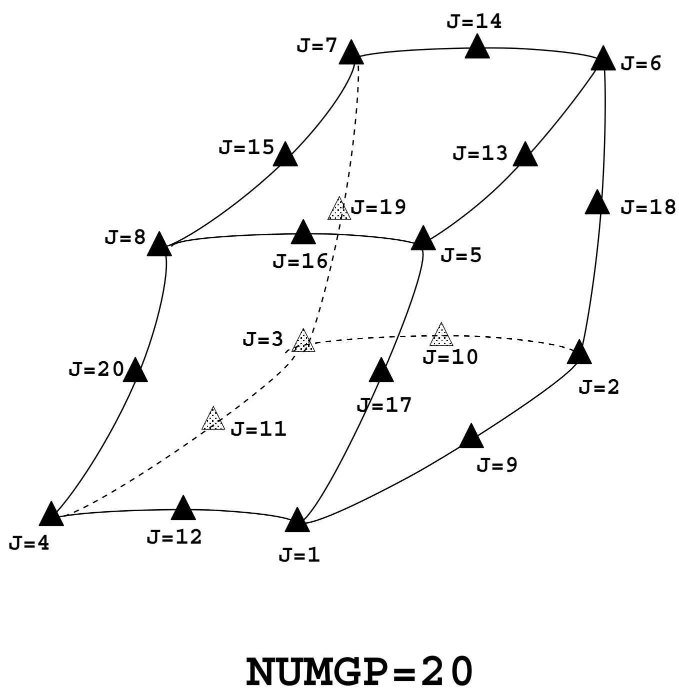


Figure 4.6: Triquadratic generation of nodal points over a volume.

4.3 Generation Point Coordinate Lines

These data lines are part of a generation sequence, and have the format 2i5,NSD*(ISHELL+1)f16. The coordinates of each generation point are defined by a generation point coordinate line. The generation points must be read in order (J=2,3,...,NUMGP) following the nodal coordinate line (J=1) which initiated the generation sequence. A nodal increment line (Section 4.4) follows the last generation line (J=NUMGP) of the sequence.

Columns	Variable	Description
1- 5	M	Node number ($1 \leq M \leq NUMNP$)
6- 10	MGEN	Set to 0
11- 26	x(1,M)	x_1 coordinate of node M(x_1^+ for shell nodes)
27- 42	x(2,M)	x_2 coordinate of node M(x_2^+ for shell nodes)
43- 58	x(3,M)	x_3 coordinate of node M(x_3^+ for shell nodes)
59- 74	x(4,M)	x_1 coordinate of node M(x_1^- for shell nodes)
75- 90	x(5,M)	x_2 coordinate of node M(x_2^- for shell nodes)
91-106	x(6,M)	x_3 coordinate of node M(x_3^- for shell nodes)

4.4 Nodal Increment Lines

Each nodal generation sequence is terminated with a nodal increment line having the format 6i5.

Columns	Variable	Description
1- 5	NINC(1)	Number of nodal increments for direction 1 ($\geq$ 0)
6-10	INC(1)	Node number increment for direction 1
11-15	NINC(2)	Number of nodal increments for direction 2 ($\geq$ 0)
16-20	INC(2)	Node number increment for direction 2
21-25	NINC(3)	Number of nodal increments for direction 3 ($\geq$ 0)
26-30	INC(3)	Node number increment for direction 3

4.5 Termination of Nodal Coordinate Input

The input of nodal coordinate data is terminated by a blank line in the data file.

5. NODAL RESTRAINT DATA (KW = RESTRAIN)

5.1 Keyword

The keyword which initiates the input of nodal restraint data is **RESTRAIN**.

5.2 Prescribed Nodal Restraints

Nodal restraint data must be input or generated for each node which has one or more prescribed degrees of freedom. If prescribed, the nodal degree of freedom is assumed to be zero (0.0) unless it is assigned a nonzero value in Section 6.0. If one or more lines of nodal restraint data are entered for a given node, the last one read takes precedence. The format for each prescribed nodal data line is (3+NDOF)*i5 with the following breakdown of information on each line:

Columns	Variable	Description
1- 5	N	Node number ($1 \leq N \leq NUMNP$)
6-10	IDUM	Set to 0
11-15	ID(1,N)	DOF 1 code for node N
16-20	ID(2,N)	DOF 2 code for node N
	.	.
	etc.	.
	.	.
	ID(NDOF,N)	DOF NDOF code for node N
	IGEN	If 0, do not initiate generation
		If 1, 1-D generation
		If 2, 2-D generation
		If 3, 3-D generation

When IGEN is nonzero, nodal restraint data is generated by a two-line sequence as follows:

Line 1: N,IDUM,ID(1,N),...,ID(NDOF,N)

Line 2: NINC(1),INC(1),NINC(2),INC(2),NINC(3),INC(3)

where NINC(1),INC(1) represent the number of nodal increments and the nodal increment, respectively in direction 1, NINC(2),INC(2) represent the number of nodal increments and the nodal increment, respectively in direction 2, and NINC(3),INC(3) represent the number of nodal increments and the nodal increment, respectively in direction 3.

5.3 Termination of Nodal Restrain Data

The input of nodal restraint data is terminated with a blank line.

6. NODAL LOAD CASES (KW = LOADCASE)

6.1 Keyword

Nodal load case data in the data file is expected to follow on the line immediately after the keyword **LOADCASE**.

6.2 Description of Load Cases

Applied nodal forces/displacements/velocities/accelerations are defined through expansions of the form:

$$\mathbf{F}(I,t) = \sum_{J=1}^{NLC} \mathbf{G}_J(I) H_J(t) \qquad \text{for } I = 1, NUMNP$$

where $\mathbf{F}(I,t)$ represents either applied nodal forces or applied nodal kinematic vectors at node I at time t in the solution process, $\mathbf{G}_J(I)$ represents the vector value of the J^{th} load case at node I, and $H_J(t)$ represents the scalar value of the J^{th} load-time function at time t. Hence the J^{th} load case is modulated in time by the J^{th} load-time function. For a given data set, the number of load cases is set by the global input variable NLC on the second control line (Section 2.0). The input data for each load case has the following sequence

- Load Case Line
- Load Case Data
- A single blank line to terminate input of this case.

6.3 Load Case Line

There can be up to NLC load cases entered in the data set and the load cases need not be entered sequentially. Hence this line tells the program which load case to expect immediately afterward. The format for this line is i5.

Columns	Variable	Description
1- 5	J	Load case number ($1 \leq J \leq NLC$)

6.4 Load Case Data

If no data is put in for a given load case, then that load case has a value of zero (0.0) at all nodes (*i.e.* $G_J(I) = 0$ for I=1,NUMNP). Load case data for the J^{th} load case begins immediately after the load case line. The format for a given nodal force or displacement line of data is 2i5,NDOF*f16.

Columns	Variable	Description
1- 5	N	Node number ($1 \leq N \leq NUMNP$)
6- 10	NUMGP	Number of generation points (≥ 0)
		if 0, no generation lines follow
		if > 0, NUMGP-1 generation lines follow
11- 26	G(1,N,J)	DOF 1 component at node N for loadcase J
27- 42	G(2,N,J)	DOF 2 component at node N for loadcase J
43- 58	.	.
59- 74	etc.	.
75- 90	.	.
91-106	G(NDOF,N,J)	DOF component at node N for loadcase J

If NUMGP is greater than zero, then an isoparametric data generation sequence the same as that used for coordinate data and initial displacement/velocity data is employed (See Section 4.0, for example). Lines 2 to NUMGP of the generation sequence define the applied forces/displacements/ velocities/ accelerations for the generation nodal points. The final line of the sequence defines the nodal increments (See Section 4.4). After the generation sequence is completed, additional lines of data may follow.

The generation may be performed along a line or curve, over a surface, or over a volume.

6.5 Load Case Generation Point Data Lines

When the input parameter NUMGP is nonzero, a multi-line generation sequence is employed to generate load case data. The format of the generation lines is 2i5,NDOF*f16.

Columns	Variable	Description
1- 5	M	Node number ($1 \leq M \leq NUMNP$)
6- 10	MGEN	Set to 0
11- 26	G(1,M,J)	DOF 1 component at node M for loadcase J
27- 42	G(2,M,J)	DOF 2 component at node M for loadcase J
43- 58	.	.
59- 74	.	etc.
75- 90	.	.
91-106	G(NDOF,M,J)	DOF NDOF component at node M for loadcase J

6.6 Nodal Increment Lines

Each nodal load case generation sequence is terminated with a nodal increment line having the format 6i5.

Columns	Variable	Description
1- 5	NINC(1)	Number of nodal increments for direction 1 (≥ 0)
6-10	INC(1)	Node number increment for direction 1
11-15	NINC(2)	Number of nodal increments for direction 2 (≥ 0)
16-20	INC(2)	Node number increment for direction 2
21-25	NINC(3)	Number of nodal increments for direction 3 (≥ 0)
26-30	INC(3)	Node number increment for direction 3

6.7 Termination of Load Case Input Data

The input of each load case is terminated by a blank line in the data file. Following the last load case, an additional blank line is entered in the data set.

7. LOAD-TIME FUNCTIONS (KW = LOADTIME)

7.1 Keyword

The input of load-time data commences immediately following the keyword **LOADTIME** in the data set.

7.2 Description of Load-Time Functions

Each data set must have at least one load-time function, and each load-time function H_J(t) is typically defined by (NLSN+1) pairs of time instants and function values. (An exception to this rule are sinusoidal load-time functions which can be entered in a simplified manner as described in Section 7.6.) Note that NLSN must be less than or equal to the maximum number of load steps permissible, which is specified by the input variable NLS on the second initial control line of the data set. A schematic of a typical load-time function H_J(t) is shown in Figure 7.1. The time instants must be in ascending order, and time intervals need not be constant. As is shown in Figure 7.1, the load-time function is assumed to behave in a piecewise linear fashion between data points. For values of t outside of the interval $[t_1 - t_{NLSN}]$, the H_J values are specified by constant extrapolation. That is, for $t < t_1$, $H_J(t) = H_J(t_1)$, and for $t > t_{NLSN}$, $H_J(t) = H_J(t_{NLSN})$. As an example of this feature, we may take NLC=1, and have the load time function value H_J(t) constant throughout the analysis. For this case, we could set $NLS = 0$ on the second initial control line and simply read in one data point to define H_J(t).

In addition to modulating the load-cases in time, load-time functions are used to modulate element consistent loads such as gravity and surface tractions. Load-time functions can also be used to vary the time step used in analysis.

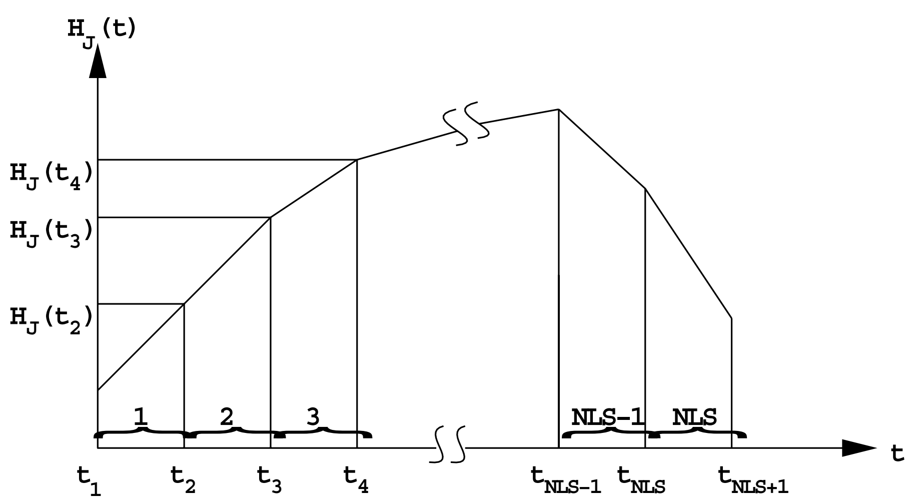


Figure 7.1: Schematic of a load-time function H_J(t).

7.3 Initial Line of Load-Time Function

There can be many load-time functions in a data file, and the load-time functions need not be read in order. Each load-time function does, however, have a number N associated with it. The format for the initial line of load-time function data is 2i5,2f12, with a description of the input variables as follows:

Columns	Variable	Description
1- 5	N	Load-time function number
6-10	NLSN	Number of load steps for this function ($\leq$NLS)
		(if $>$ 0, then NLSN data lines follow to
		complete the definition of this function.
11-22	H_N(1,1)	Time instant 1 (*i.e.* t_1).
23-34	H_N(1,2)	Function value at t_1.

7.4 Subsequent Input Lines for a Load-Time Function

If NLSN is greater than zero, then NLSN additional lines of data will be required to complete the input of load-time function N. The format for these subsequent lines is 10X,2f12.

Columns	Variable	Description
11-22	H_N(J,1)	J^{th} time instant (*i.e.* t_j)
23-34	H_N(J,2)	Function value at t_j.

7.5 Transition Between Load-Time Functions

There should be no blank lines between successive load-time functions in a data file.

7.6 Simple Input of Sinusoidal Load-Time Functions

Sinusoidal load-time functions can be entered easily by using negative values for load-time function N values (Section 7.3). When a negative value is read, FENDAC expects on the next line of data in **10x,5f12** format, entry of values A, ϕ, and ω. Since the N^{th} load-time function will thus have the form $F_N(t) = A\sin(\omega t + \phi)$, A denotes the amplitude of the function, ϕ the starting phase angle, and ω the radian frequency.

7.7 Termination of Load-Time Function Input

After entry of the last load-time function, terminate input of load-time function data with one or more blank lines in the data file.

8. TYPE-1 NODAL ENSLAVEMENTS (KW = SLAVENOD)

8.1 Keyword

The input of nodal enslavement data commences immediately following the keyword **SLAVENOD** in the data set.

8.2 Description of Type-1 Nodal Enslavements

It is often desirable to force two or more nodes in a FEM mesh to share identical nodal degrees of freedom. One way of achieving this objective (nodal link elements could also be used) is through nodal enslavements, in which a node is specified as a master node, for a corresponding slave node. Optionally, either all degrees of freedom or only a subset thereof associated with a slave node can be tied to those of the master node. The format for input of nodal enslavement data is described below.

8.3 Nodal Enslavement Data

FENDAC expects the user to input data for NSLAVE nodes. This data can be entered either in NSLAVE lines of data if no generation is used, or fewer than NSLAVE lines of data if data generation procedures are employed. The input data format for each line of data is **9i5,** with the data having the following significance:

Columns	Variable	Description
1- 5	ISLAVE(N, 1)	Master node number
6-10	ISLAVE(N, 2)	Slave node number
11-15	ISLAVE(N, 3)	Enslavement code for DOF 1
		= 0: no enslavement
		= 1: enslavement
16-20	ISLAVE(N, 4)	Enslavement code for DOF 2
		= 0: no enslavement
		= 1: enslavement
⋮	⋮	⋮
	ISLAVE(N,NDOF)	Enslavement code for DOF NDOF
		= 0: no enslavement
		= 1: enslavement
	IGEN	Data generation parameter
		= 0: no data generation
		= 1: 1-D generation
		= 2: 2-D generation
		= 3: 3-D generation

When IGEN is nonzero, nodal enslavement data is generated by providing a second line of data as follows having the format **6i5.**

Line 2: NINC(1),INC(1),NINC(2),INC(2),NINC(3),INC(3)

where NINC(1),INC(1) represent the number of nodal increments and the nodal increment, respectively in direction 1, NINC(2),INC(2) represent the number of nodal increments and the nodal increment, respectively in direction 2, and NINC(3),INC(3) represent the number of nodal increments and the nodal increment, respectively in direction 3.

8.4 Termination of Nodal Enslavement Data

The input of nodal enslavement data is terminated with a blank line.

9. TYPE-2 NODAL ENSLAVEMENTS (KW = SLAV2NOD)

9.1 Keyword

The input of nodal enslavement data commences immediately following the keyword **SLAV2NOD** in the data set.

9.2 Description of Type-2 Nodal Enslavements

When adaptive mesh refinement is used, FEM models such as that in Figure 9.1 will often occur. Assuming that the elements shown use bilinear interpolation, a constraint on certain nodal displacements is needed in order to ensure continuity of the displacement field across element boundaries. As an example, for the mesh in Figure 9.1, it would be required that $\mathbf{u}_B = \frac{1}{2}\mathbf{u}_A + \frac{1}{2}\mathbf{u}_C$ and also $\mathbf{u}_G = \frac{1}{2}\mathbf{u}_D + \frac{1}{2}\mathbf{u}_J$ In this case, we would say that Node B has a **Type-2** enslavement relationship with Nodes A and C, and that Node G has a similar enslavement relation with Nodes D and J. The format for input of Type-2 nodal enslavement data is described below.

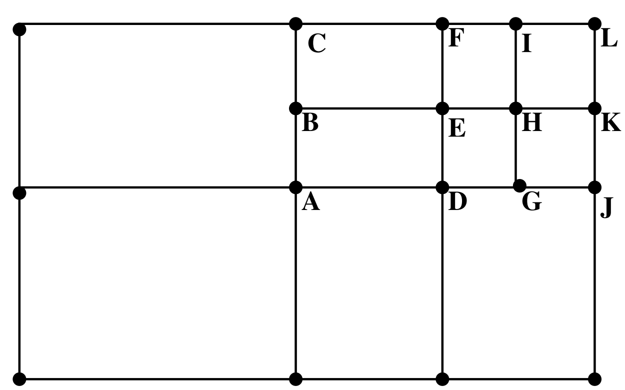


Figure 9.1: Schematic of a mesh in which node B would need to have a type-2 enslavement relation with nodes A and C, and similarly, node G would have type-2 enslavement to nodes D and J.

9.3 Type-2 Nodal Enslavement Data

FENDAC expects the user to input Type-2 enslavement data for NSLAVE2 nodes. Since no generation is presently available in this module, the data must be entered on NSLAVE2 separate lines. Three integer node number values are provided on each line of data, one for the number of the enslaved node, and two for the numbers of the master nodes. The input data format for each line of data is **3i10**.

Columns	Variable	Description
1-10	ISLAVE2(N,1)	Enslaved node number
11-20	ISLAVE2(N,2)	First master node number
21-30	ISLAVE2(N,3)	Second master node number

9.4 Termination of Type-2 Nodal Enslavement Data

The input of nodal enslavement data is terminated with a blank line.

10. NODAL TIME HISTORIES (KW = NODEHIST)

10.1 Keyword

The input of nodal time history data commences immediately following the keyword **NODEHIST** in the data set.

10.2 Nodal Time History Data

FENDAC expects the user to input data for NHIST nodal time histories. Since no data generation is available with this input module, this data must be entered on NHIST lines of data having the format **3i5** and variable descriptions:

Columns	Variable	Description
1- 5	NODE	Node number
6-10	IDOF	Degree of freedom number
11-15	IDVA	Force/Displacement/Velocity/Acceleration Indicator
		= 1: displacement
		= 2: velocity (ITYPE $\geq$ 2)
		= 3: acceleration (ITYPE = 3)
		= -1: internal force

10.3 Termination of Nodal Time History Data

The input of nodal restraint data is terminated with a blank line.

10.4 Homogenization Time Histories

When the control variable IAVHIST is set to 1, FENDAC automatically generates in addition to requested nodal time histories, time histories of volume-averaged stress $\boldsymbol{\Sigma}$ and volume-averaged strain **E** in the solid continuum element groups. The user need not reset NHIST to take account of these additional histories.

11. PERIODIC BOUNDARY CONDITIONS (KW = PERIODIC)

11.1 Keyword

The input of periodic boundary condition data commences immediately following the keyword **PERIODIC** in the data set.

11.2 Usage of Periodic Boundary Condition Data

This feature is used with mechanical homogenization of periodic and/or quasi-periodic materials. When performing homogenization computations, either stress-controlled or strain-controlled methods can be employed[1]. For stress-controlled homogenization, FENDAC uses compatible surface elements (Sections 13.5 and 13.6) and prescribed surface tractions (specified in element group data). For strain-controlled homogenization (IBOUND < 0), FENDAC uses applied macroscopic strain tensors **E** and nodal enslavements (Section 8). This section describes how to prescribe macroscopic strain tensors **E** in the context of strain-controlled homogenization.

A symmetric macroscopic strain tensor **E** having six distinct components is imposed on the domain. Each component of **E** is assigned a load time function to control its variation in time. This data is put in on six lines having free format and the sequence shown below:

Line Number	Columns	Variable	Description
1	arbitrary	ESTR(1)	$\mathbf{E}_{11}$
	arbitrary	ISTRAIN(1)	LTF controlling $\mathbf{E}_{11}$
2	arbitrary	ESTR(2)	$\mathbf{E}_{22}$
	arbitrary	ISTRAIN(2)	LTF controlling $\mathbf{E}_{22}$
3	arbitrary	ESTR(3)	$\mathbf{E}_{33}$
	arbitrary	ISTRAIN(3)	LTF controlling $\mathbf{E}_{33}$
4	arbitrary	ESTR(4)	$\mathbf{E}_{23}$
	arbitrary	ISTRAIN(4)	LTF controlling $\mathbf{E}_{23}$
5	arbitrary	ESTR(5)	$\mathbf{E}_{31}$
	arbitrary	ISTRAIN(5)	LTF controlling $\mathbf{E}_{32}$
6	arbitrary	ESTR(6)	$\mathbf{E}_{12}$
	arbitrary	ISTRAIN(6)	LTF controlling $\mathbf{E}_{12}$

Terminate input of periodic data with a blank line.

12. TOPOLOGY OPTIMIZATION CONTROL DATA (KW = TOPOLOGY)

12.1 Keyword

The input of topology optimization data commences immediately following the keyword **TOPOLOGY** in the data set. This data will be read only when the global parameter IDESIGN is set to an integer value larger than 0 on the master control line of the data file. The master control line data is documented in Section 2 of this manual.

12.2 What does topology optimization do?

Topology optimization is used to find the optimal distribution of materials to carry loads applied to a body. FENDAC has the capability to perform topology optimization both for structures and at reduced length scales, for composite materials. As an example, Figure 12.1 below shows the initial and final distributions of materials in a two-material graphite-epoxy aligned fiber composite. For composite materials, topology optimization redistributes the materials present in a manner that optimizes specific mechanical performance properties of the composite. Further examples of topology optimization of material structure in composite materials is presented in [7], while numerous examples and background on topological optimization of structural systems is described in [8-11].

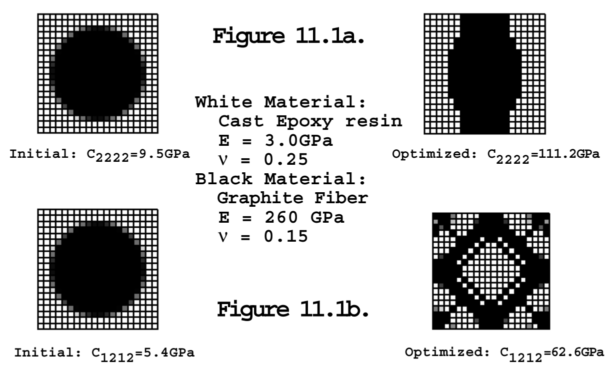


Figure 12.1 Unit cells of initial and optimized graphite-epoxy macrostructures. Figure 11.1a shows macrostructure optimizing C_{2222} with $\Psi = |E_{22}|$ under applied S_{22} loading. Figure 11.1b shows macrostructure optimizing C_{1212} with $\Psi = |E_{12}|$ under applied S_{12} loading.

12.3 How topology optimization is performed

To perform topology optimization, the optimization problem must be defined by a number of functionals. Generally one of the functionals will be the so-called objective functional whose value is to be minimized and the remaining functionals will be so-called constraint functionals. An optimization program such as IDESIGN or LINDO is always required to manage the optimization computations. Each time the optimization program needs to evaluate the functionals or compute their design gradients, the program calls FENDAC to do this job. FENDAC then writes the information in a number of binary files that the optimization program can read. In fact then, FENDAC is executed repeatedly and automatically by an optimization program during topological optimization.

12.4 Expected format for input data

FENDAC expects to read in descriptive data for one **objective** functional, and descriptive data for up to ten **constraint** functionals. The first functional read in is taken to be the **objective** functional and subsequent ones are treated as constraints. To tell FENDAC how many constraint functionals will be used in defining the optimization problem, the user must first input the variable NFUNCT $\leq$ 10 on a separate line, followed by (NFUNCT+1) lines of data describing the prescribed functionals.

12.4.1 Input of control parameter NFUNCT

The expected format for input of the parameter NFUNCT is **i5** on a separate line immediately following the keyword **TOPOLOGY**.

12.4.2 Specifying the functionals

After reading in the parameter NFUNCT, FENDAC then expects to read in the appropriate number of lines of data, each having the format (**2I5,2F10**). There must be one line of data for each functional used in the optimization problem. The required variables have the following meanings:

Columns	Variable	Description
1- 5	IFUNC(I,1)	Primary functional descriptor
		If 1, global volume fraction functional
		If 2, global strain energy functional
		If 3, global eigenvalue functional
		If 4, macro stress-strain functional
		If 5, displacement norm functional
		If 6, "perimeter" functional
6-10	IFUNC(I,2)	Secondary functional descriptor
		Consult Table 12.2
11-20	CVAL(I)	Scalar constant which is subtracted from
		the computed functional value
21-30	SVAL(I)	Scalar constant by which the computed functional
		value is multiplied.

Table 12.1 Functional data employed in topology optimization.

FENDAC uses the secondary functional descriptors to provide additional information for each functional. The meaning and usage of the secondary descriptors is as follows:

PRIMARY DESCRIPTOR IFUNCT(I,1)	TYPE OF FUNCTIONAL	SECONDARY DESCRIPTOR IFUNCT(I,2)
1	Global volume fractions	If 1: $\mathcal{F}_I =< \phi >$
		If 2: $\mathcal{F}_I =< \phi > \cdot < 1 - \phi >$
2	Global strain energy	If 1: Force-controlled
		If 2: Displacement-controlled
3	Global eigenvalues	Gives the eigenvalue number
4	Macro stresses and strains	If 1: $\mathcal{F}_I = E_{11}$
		If 2: $\mathcal{F}_I = E_{22}$
		If 3: $\mathcal{F}_I = E_{33}$
		If 4: $\mathcal{F}_I = 2E_{23}$
		If 5: $\mathcal{F}_I = 2E_{31}$
		If 6: $\mathcal{F}_I = 2E_{12}$
		If 7: $\mathcal{F}_I = S_{11}$
		If 8: $\mathcal{F}_I = S_{22}$
		If 9: $\mathcal{F}_I = S_{33}$
		If 10: $\mathcal{F}_I = S_{23}$
		If 11: $\mathcal{F}_I = S_{31}$
		If 12: $\mathcal{F}_I = S_{12}$
	Poisson's ratio functional	If 13: $\mathcal{F}_I = \frac{-E_{11}}{E_{22}}$
	Poisson's ratio functional	If 14: $\mathcal{F}_I = \frac{-E_{22}}{E_{33}}$
	Poisson's ratio functional	If 15: $\mathcal{F}_I = \frac{-E_{33}}{E_{11}}$
	Strain-controlled energy	If 16: $\mathcal{F}_I = \int \mathbf{S} : \dot{\mathbf{E}} d\tau$
	Stress-controlled energy	If 17: $\mathcal{F}_I = \int \mathbf{S} : \dot{\mathbf{E}} d\tau$
5	Displacement norm functional	Specifies mode shape or load case number defining the nodes.
6	Perimeter control functional	Unused.

Table 12.2 Secondary functional descriptors.

13. INITIAL DISPLACEMENT DATA (KW = INITIALD)

13.1 Keyword

The keyword for this input module is **INITIALD.**

13.2 Nodal Displacement Lines

For parabolic and hyperbolic BVPs, an initial displacement field can be specified. If no initial displacement data is entered, the initial field is homogeneous. Input of the initial displacement data is much like that of nodal coordinate data. Initial displacement data begins on the line immediately following the keyword **INITIALD**. The first line is always a Nodal Displacement Line having the format (2i5,NDOF*f16).

Columns	Variable	Description
1- 5	N	Node number ($1 \leq N \leq NUMNP$)
6-10	NUMGP	Number of generation points (≥ 0)
		if 0, no generation lines follow
		if > 0, NUMGP-1 generation lines follow
11- 26	D(1,N)	DOF1 displacement of node N
27- 42	D(2,N)	DOF2 displacement of node N
43- 58	D(3,N)	DOF3 displacement of node N
59- 74	D(4,N)	DOF4 displacement of node N
75- 90	D(5,N)	DOF5 displacement of node N
91-106	D(6,N)	DOF6 displacement of node N

Notes:

1. If NUMGP is greater than zero, this data line initiates an isoparametric data generation sequence. Lines 2 to NUMGP following the nodal displacement line define the displacements of the additional generation points (Section 6.5). The final data line of the generation sequence is the nodal increment line (Section 6.6). After a generation sequence is completed, additional nodal displacement lines or generation sequences may follow.
2. The generation may be performed along a line or curve (for NSD = 1,2,3), over a surface (for NSD = 2,3), or over a volume (for NSD = 3). A description of each of these options was provided in Section 4.

13.3 Generation Point Displacements

These data lines are part of a generation sequence, and have the format **2i5,NDOF*f16**. The displacement at each generation point is defined by a generation point displacement line. The generation points must be read in order (J=2,3,..., NUMGP) following the nodal displacement line (J=1) which initiated the generation sequence. A nodal increment line follows the last generation line (J=NUMGP) of the sequence.

Columns	Variable	Description
1- 5	M	Node number ($1 \leq M \leq NUMNP$)
6- 10	MGEN	Set to 0
11- 26	D(1,M)	DOF1 displacement of node M
27- 42	D(2,M)	DOF2 displacement of node M
43- 58	D(3,M)	DOF3 displacement of node M
59- 74	D(4,M)	DOF4 displacement of node M
75- 90	D(5,M)	DOF5 displacement of node M
91-106	D(6,M)	DOF6 displacement of node M

13.4 Nodal Increment Lines

Each nodal generation sequence is terminated with a nodal increment line having the format 6i5.

Columns	Variable	Description
1- 5	NINC(1)	Number of nodal increments for direction 1 (≥ 0)
6-10	INC(1)	Node number increment for direction 1
11-15	NINC(2)	Number of nodal increments for direction 2 (≥ 0)
16-20	INC(2)	Node number increment for direction 2
21-25	NINC(3)	Number of nodal increments for direction 3 (≥ 0)
26-30	INC(3)	Node number increment for direction 3

13.5 Termination of Nodal Displacement Input

The input of nodal displacement data is terminated by a blank line.

14. PURELY HYPERBOLIC PROBLEMS

14.1 Initial Velocity Field (KW = INITIALV)

For hyperbolic initial value-boundary value problems, the user is permitted to specify both an initial displacement field and an initial velocity field. The process of entering the data associated with both fields is virtually identical. Input of the initial displacement field was described in Section 12. Input of the initial velocity field is completely analogous to that of the initial displacement field with the exception that the keyword is different.

14.1.1 Keyword

The keyword for this input module is **INITIALV.** The initial velocity field data begins in the data file on the line immediately following that on which the keyword **INITIALV** occurs. If no data follows the keyword, then FENDAC creates a homogeneous initial velocity field with $\boldsymbol{v} = \mathbf{o}$, uniformly.

14.2 Rayleigh Damping Parameters (KW = RAYLDAMP)

For structural dynamics type problems, it is often desirable to employ Rayleigh damping. FENDAC allows the user to input different damping quantities for each element group, thus allowing different parts of a structure to have different damping characteristics. The input of Rayleigh damping data begins in the input data file immediately following the keyword **RAYLDAMP**. For each element group a single line of data having the format **3f10** is expected. The data on each line has the significance:

Columns	Variable	Description
1-10	ξ_k	Damping ratio for kth element group
11-20	ω_{k1}	Estimated frequency of 1st mode
21-30	ω_{k2}	Estimated frequency of 2nd mode

The damping data for each element group is expected to appear sequentially in the data file. FENDAC expects to find NUMEG lines of data, where NUMEG is the number of element groups specified for a given problem.

Terminate input of Rayleigh Damping data with a blank line.

14.3 Excitation Stabilization (KW = STABILIZ)

For hyperbolic initial value-boundary value problems in which the essential boundary conditions are provided through discretized acceleration records, drifting instabilities can arise due to the numerical integration of the acceleration record to obtain velocity and displacement histories. To alleviate this problem, which occurs only when ITYPE=3, and LDVA=3, drift stabilization parameters can be communicated to FENDAC. The drift stabilization parameters follow the occurance of the keyword **STABILIZ**. Iff ITYPE=3 and LDVA=3, then FENDAC expects drift stabilization data for each of the NLTF load time functions. The stabilization parameters for each load time function are entered in free format as follows:

Columns	Variable	Description
arbitrary	V_{k1}	Linear velocity time corrector
arbitrary	V_{k2}	Velocity zero-corrector
arbitrary	D_{k1}	Linear displacement time corrector
arbitrary	D_{k2}	Displacement zero-corrector
arbitrary	NTSTAB$_k$	Time step at which zero corrections are applied

As implemented, the drift stabilized algorithms for computing displacements at essential boundary nodes is

$$d_{n+1} = d_n + v_n \Delta t + \frac{1}{2}(1 - 2\beta)(\Delta t)^2 a_n + \beta(\Delta t)^2 a_{n+1} - \sum_{k=1}^{NLTF} D_{k1}\Delta t,$$

while the stabilized algorithm for computing velocities at essential boundary nodes is

$$v_{n+1} = v_n + a_n \alpha \Delta t + (1 - \alpha)(\Delta t) a_{n+1} - \sum_{k=1}^{NLTF} V_{k1}\Delta t.$$

At time $t = t_{NSTAB_k}$, zero-corrections are applied to essential displacements and velocities by the relations

$$d = d - D_{k2}$$
$$v = v - V_{k2}.$$

It should be noted that to be fully effective, the input drift stabilization parameters are dependent upon the Newmark integration parameters employed.

Terminate input of stabilization data with a blank line.

15. EIGENVALUE SOLVER KW = EIGENSOL

15.1 Background

The eigenvalue solver in FENDAC is used to solve for free vibrational modes of vibration of structural/continuum systems as follows:

$$(\mathbf{K} - \lambda_k \mathbf{M}) \cdot \mathbf{y}_k = \mathbf{0},$$

where **K** is the stiffness matrix of the system, **M** is the mass matrix, λ_k is the k^{th} eigenvalue of the system, and $\mathbf{y}_k$ is the k^{th} eigenvector of the system. As many eigenvalues of the above system can be obtained as desired, although only the first few are typically ever used, and the higher mode eigenvalues are increasingly difficult to compute accurately. Eigenvalues can be obtained using either a diagonalized "lumped" mass, or the "consistent" mass matrix.

15.2 Required Input

Following the occurance of the keyword **EIGENSOL** in the data file, the user simply needs to enter in **I5** format the value of *NFIND* which specifies the number of eigenvalues and eigenvectors being requested from the analysis. (Note: the algorithm employed always finds the lowest eigenmodes first and proceeds toward increasingly higher modes.)

16. ELEMENT TYPE DOCUMENTATION

General Input of Element Data

Element data should be arranged to come last in the data file. There may be many element groups within a given data file, and numbers are assigned to groups corresponding to the order in which they are arranged in the data file. The keyword that should precede input for each element group's data is **ELMNTGRP**.

Element types available in FENDAC and described in this section are listed below:

AVAILABLE ELEMENT TYPES

16.1 2-D BILINEAR LAPLACIAN ELEMENT

This element may be used in either quadrilateral (4-node) or triangular (3-node) mode to solve the planar form of the general Laplace equation:

$$\nabla \cdot \mathbf{v} = g,$$

in which

$$\mathbf{v} = -\mathbf{k} \cdot (\nabla h).$$

In the preceding expression, **k** denotes a conductivity or permeability tensor, g a source/sink function, and h a potential function (either temperature or piezometric head, for example). The plane of analysis is the x_1-x_2 plane, and the element is assumed to have a unit thickness in the out-of-plane direction. To use this element, the global control variables must be set such that NSD=2 and NDOF=1 (See Section 2.0).

Input for this element is comprised of three segments:

1. A master control line,
2. Input of material properties (*i.e.* **k**, ρ, and g), and
3. Input of element connectivities.

16.1.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 4
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)

16.1.2 Material Property Data

The material property data follows immediately after the master control line, and there must be NUMAT lines of material property data, one line for each material property type. The format for each line of material property data is: I5,5X,8F10. The material properties to be entered are of the form:

Columns	Variable	Description
1- 5	MAT	Material number
11-20	k_{11}	Permeability tensor component
21-30	k_{12}	Permeability tensor component
		due to symmetry $k_{21} = k_{12}$
31-40	k_{22}	Permeability tensor component
41-50	g	Source/sink term
51-60	ρ	Material mass density

16.1.3 Element Connectivity Data

The format for element connectivity data is 7I5.

Columns	Variable	Description
1- 5	N	Element number ($0 < N \leq NUMEL$)
6-10	MAT(N)	Material set number (≥ 0)
11-15	IEN(1,N)	Number of 1^{st} node
16-20	IEN(2,N)	Number of 2^{nd} node
21-25	IEN(3,N)	Number of 3^{rd} node
26-30	IEN(4,N)	Number of 4^{th} node
31-35	NG	Generation parameter
		if 0, no generation
		if ≥ 1, generate element data

16.1.4 Generation Data Input

See Figure 16.1 for a schematic representation of the generation scheme. The input format for each line is 6I5.

Columns	Variable	Description
1- 5	NEL(1)	Number of elements in direction 1 ≥ 0; if = 0, set internally to 1
6-10	INCEL(1)	Element number increment for direction 1 if = 0, set internally to 1
11-15	INC(1)	Node number increment for direction 1 if = 0, set internally to 1
16-20	NEL(2)	Number of elements in direction 2 ≥ 0; if = 0, set internally to 1
21-25	INCEL(2)	Element number increment for direction 2 if = 0, set internally to 1
26-30	INC(2)	Node number increment for direction 2 if = 0, set internally to 1

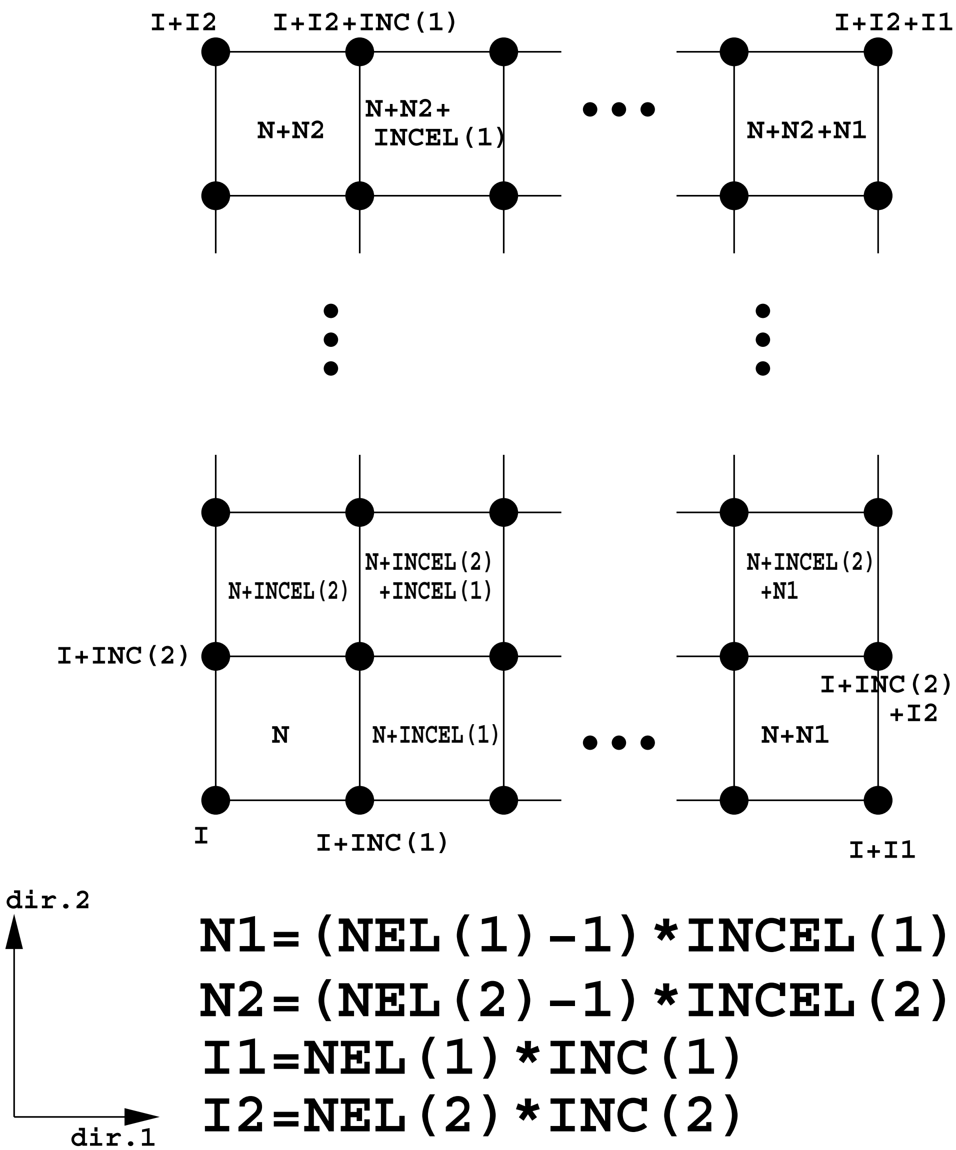


Figure 16.1: Schematic of element data generation for 4-node bilinear elements.

16.2 2-D BILINEAR CONTINUUM ELEMENT

This element may be used in either quadrilateral (4-node) or triangular (3-node) mode to solve the planar or cylindrical forms of the momentum balance equation:

$$\nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{g} = \rho \mathbf{a},$$

in which $\boldsymbol{\sigma}(\epsilon)$ is given by any one of a number of different stress-strain constitutive models. In the preceding expression, $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor, $\mathbf{g}$ denotes a gravitational body force vector, ρ the material mass density, and $\mathbf{a}$ the particle acceleration vector. This element can be used to solve either quasi-static problems (elliptic BVPs) or dynamic problems (hyperbolic BVPs).

Input for this element is comprised of up to six segments:

1. A master control line,
2. Input of material properties,
3. Element gravity vectors
4. Input of element connectivities,
5. Applied surface tractions (if used), and
6. Element time histories.

16.2.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 3
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)
16-20	MTYPE	Material Type
21-25	IOPT	Analysis Option
		IOPT=0, plane strain analysis
		IOPT=1, plane stress analysis
		IOPT=2, cylindrically symmetric
26-30	IFD	Finite Deformation Option
		IFD = 0, Neglect finite deformations
		IFD = 1, Account for finite deformations
31-35	NSURF	Number of surface tractions to be applied
		Surface Traction Follows Connectivity Data
36-40	IBBAR	Strain-Displacement Option
		IBBAR=0, Standard Formulation
		IBBAR=1, Mean-Dilatational Formulation
41-45	NHIST	Number of element time histories desired
		Element time history data follows element
		connectivity and surface traction data
46-50	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
		= 2: Printout at element quadrature points
51-55	LCASG	Load-time function modulating element gravity
56-60	LCASP	Load-time function modulating element normal tractions
61-65	LCASS	Load-time function modulating element shear tractions
66-70	ITAN	Continuum/Consistent Tangent Option
		= 0: Continuum Tangent Operators
		= 1: Consistent Tangent Operators
71-75	NQUAD	Number of quadrature points per element (1 or 4)
76-80	IFRAC	Number of volume fractions per element

Columns	Variable	Description
81-85	ICHECK	Design variable spatial filtering option
		(Used only with continuum topology optimization)
		= 0: Do not use filtering
		= 1: Employ filtering
86-90	IMIX	Code for mixing rule usage in topology optimization
		= 0: Powerlaw rule
		= 1: Voigt-Reuss rule
91-95	IMATIN	Code for special input format of material data
		= 0: Standard input format
		= 1: Input in a separate file MATERIAL.data

16.2.2 Material Property Data

When IMATIN=0 (standard material input mode), the material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one line for each material type. The format for each line of material property data set depends upon the type of constitutive model being used. Consult Section 17 for specific models.

On the other hand, when IMATIN=1, the mode for input of material data is different. With IMATIN=1, it is assumed that each element will have its own set of material properties. As this option is available only with linear elasticity, the input format for the file MATERIAL.data is as follows.

element number, n
mass density, rho
$C_{11}, C_{12}, C_{13}, C_{14}$
$C_{21}, C_{22}, C_{23}, C_{24}$
$C_{31}, C_{32}, C_{33}, C_{34}$
$C_{41}, C_{42}, C_{43}, C_{44}$

Each line of data is format free, but there must be NUMEL sets of data, with no blank lines between sets.

16.2.3 Element Gravity Vectors

Each group of continuum elements must have a gravity vector. The input format for this element type is **2f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity
21-30	PGLOB	Parameter used in hybrid Voigt-Reuss mixing rule with topology optimization

16.2.4 Element Connectivity Data

Input of element connectivity for this element type is identical to that of the 2-D Laplacian element described in Section 16.1.3. **Terminate input of element connectivity data with a blank line.**

16.2.5 Element Surface Traction Data

Both pressure and shear tractions can be applied to the edges of quadrilateral continuum elements. Sign conventions for positive pressure tractions and positive shear tractions are shown in Figures 16.2a and 16.2b, respectively. The applied tractions can vary linearly along the sides as shown.

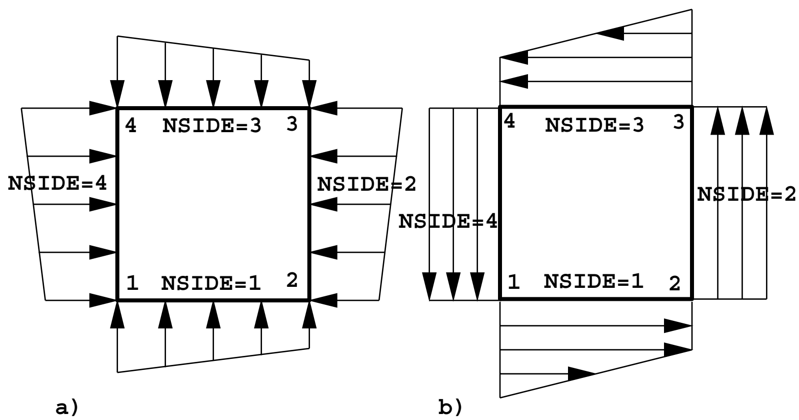


Figure 16.2: Sign conventions for applied surface pressures (a) and applied surface shears (b).

The format for input of element surface traction data is **2i5,4f10,i5,** with the following information expected

Columns	Variable	Description
1- 5	IELNUM	Element Number
6-10	NSIDE	Number of side to which traction applied
		connecting nodes 1-2: NSIDE=1
		connecting nodes 2-3: NSIDE=2
		connecting nodes 3-4: NSIDE=3
		connecting nodes 4-1: NSIDE=4
11-20	P1	left node normal traction magnitude
21-30	P2	right node normal traction magnitude
31-40	S1	left node shear traction magnitude
41-50	S2	right node shear traction magnitude
51-55	IGEN	Data Generation Parameter
		= 0: Do not use generation sequence
		= 1: Use a linear generation sequence

When IGEN = 1, then a second line of data similar to that above is entered. The first line of data corresponds to the surface tractions on the first element of a sequence, and the second line corresponds to the surface tractions on the last element of the sequence. Surface traction data is generated along intermediate elements of the sequence by interpolation. The data defining the intermediate elements is entered on a third line of data having the format **2i5:**

Columns	Variable	Description
1- 5	NINC	Number of element increments
6-10	INC	Increment of element number

Terminate input of element surface traction data with a blank line.

16.2.6 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as follows:

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired:
		= 1: σ_{11}
		= 2: σ_{22}
		= 3: σ_{12}
		= 4: σ_{33}
		= 5: $\frac{1}{3}\mathbf{tr}(\boldsymbol{\sigma})$
		= 6: $\sqrt{J_2'}$
		= 7: $\|\|\|\mathbf{s} - \mathbf{q}\|\|\|$
		= 8: ϵ_{11}
		= 9: ϵ_{22}
		=10: γ_{12}
		=11: ϵ_{33}
		=12: $\mathbf{tr}\,(\epsilon)$
		=13: $\overline{e}^p$

Terminate input of element time history data with a blank line.

16.3 3-D TRILINEAR CONTINUUM ELEMENT

This element may be used in either hexahedral (8-node), pentahedral (6-node), or tetrahedral (4-node) modes to solve 3-D forms of the momentum balance equation:

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{g} = \rho \mathbf{a},$$

in which $\boldsymbol{\sigma}(\epsilon)$ is given by any one of a number of different stress-strain constitutive models. In the preceding expression, $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor, $\mathbf{g}$ denotes a gravitational body force vector, ρ the material mass density, and $\mathbf{a}$ the particle acceleration vector. This element can be used to solve either quasi-static problems (elliptic BVPs) or dynamic problems (hyperbolic BVPs).

Input for this element is comprised of up to six segments:

1. A master control line,
2. Input of material properties,
3. Element gravity vectors
4. Input of element connectivities,
5. Applied surface tractions (if used), and
6. Element time histories.

16.3.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 6
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)
16-20	MTYPE	Material Type
21-25	ITAN	Continuum/Consistent Tangent Option
		ITAN=0, use continuum tangent
		ITAN=1, use consistent tangent
26-30	IFD	Finite Deformation Code
		IFD = 0, Neglect Finite Deformation
		IFD > 0, Include Finite Def. Effects
31-35	NSURF	Number of surface tractions to be applied
		Surface Traction Data Follows Connectivity Data
36-40	IBBAR	Strain-Displacement Option
		IBBAR=0, Standard Formulation
		IBBAR=1, Mean-Dilatational Formulation
41-45	NHIST	Number of element time histories desired
		Element time history data follows element connectivity and surface traction data
46-50	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
		= 2: Printout at element quadrature points
51-55	LCASG	Load-time function modulating element gravity
56-60	LCAST1	Load-time function modulating X_1 surface tractions
61-65	LCAST2	Load-time function modulating X_2 surface tractions
66-70	LCAST3	Load-time function modulating X_3 surface tractions
71-75	NQUAD	Number of quadrature points per element (1 or 8)
76-80	IFRAC	Number of volume fractions per element (generally used in conjunction with topology optimization.)

16.3.2 Material Property Data

The material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one line for each material type. The format for each line of material property data set depends upon the type of constitutive model being used (MTYPE). Consult Section 17 for specific material models.

16.3.3 Element Gravity Vectors

Each group of continuum elements must have a gravity vector. The input format for this element type is **3f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity
21-30	Grav(3)	X_3 component of gravity

16.3.4 Element Connectivity Data

Input of element connectivity for this element type can be used with or without data generation. Data for definition of a single element is expected in the format **11i5** in the following sequence:

Element Connectivity Data Line

Notes	Columns	Variable	Description
(1)	1- 5	N	Element number ($0 < N \leq NUMEL$)
	6-10	MAT(N)	Material set number ($\geq$ 0)
	11-15	IEN(1,N)	Number of 1^{st} node
	16-20	IEN(2,N)	Number of 2^{nd} node
	21-25	IEN(3,N)	Number of 3^{rd} node
(2)	26-30	IEN(4,N)	Number of 4^{th} node
	31-35	IEN(5,N)	Number of 5^{th} node
	36-40	IEN(6,N)	Number of 6^{th} node
	41-45	IEN(7,N)	Number of 7^{th} node
(3)	46-50	IEN(8,N)	Number of 8^{th} node
(4)	51-55	NG	Generation parameter
			if 0, no generation
			if $\geq$ 1, generate element data

Notes:

(1) All elements must be input on a nodal data line or be generated using a data generation sequence. **Terminate input of element connectivity data with a blank line.**

(2) For wedge elements (Figure 16.2), set node number IEN(4,N) equal to IEN(3,N).

(3) For wedge elements, set node number IEN(8,N) equal to IEN(7,N).

(4) If the generation parameter NG is > 0, an element data generation line is expected next in the input file.

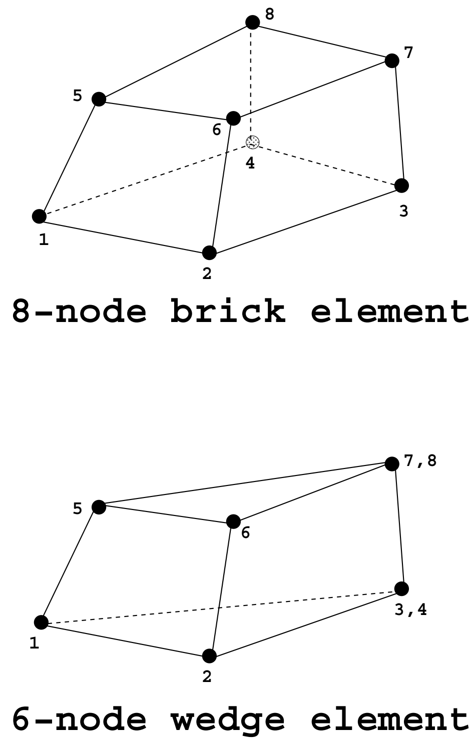
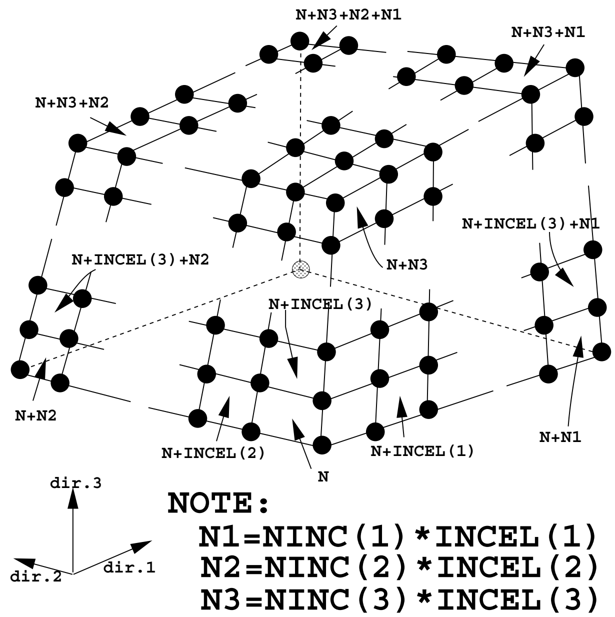


Figure 16.2: Hexahedral and pentahedral continuum elements.

16.3.5 Element Data Generation Input

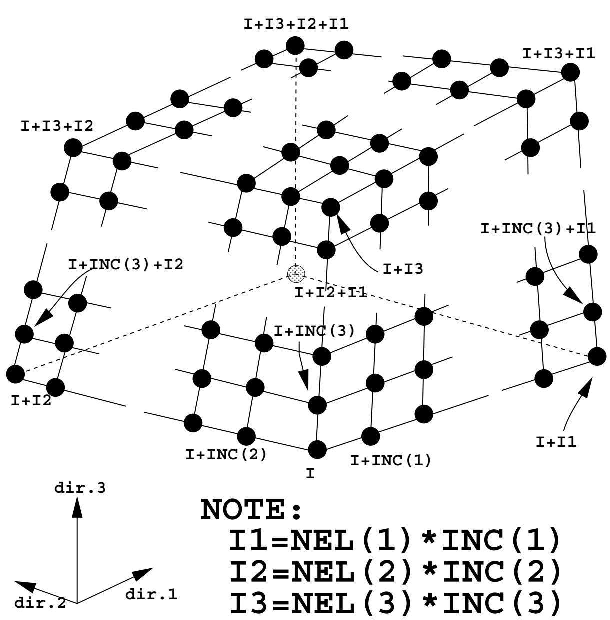
See Figure 16.3 for a schematic representation of the generation scheme. The input format such lines is **9i5.**

Columns	Variable	Description
1- 5	NEL(1)	Number of elements in direction 1 $\geq$ 0; if = 0, set internally to 1
6-10	INCEL(1)	Element number increment for direction 1 if = 0, set internally to 1
11-15	INC(1)	Node number increment for direction 1 if = 0, set internally to 1
16-20	NEL(2)	Number of elements in direction 2 $\geq$ 0; if = 0, set internally to 1
21-25	INCEL(2)	Element number increment for direction 2 if = 0, set internally to 1
26-30	INC(2)	Node number increment for direction 2 if = 0, set internally to 1
31-35	NEL(3)	Number of elements in direction 3 $\geq$ 0; if = 0, set internally to 1
36-40	INCEL(3)	Element number increment for direction 3 if = 0, set internally to 1
41-45	INC(3)	Node number increment for direction 3 if = 0, set internally to 1



ELEMENT NUMBERS

Figure 16.3a: Schematic of element number data generation for 8-node trilinear elements.



NODE NUMBERS

Figure 16.3b: Schematic of element node number data generation for 8-node trilinear elements.

16.3.6 Element Surface Traction Data

The format for input of element surface traction data is **2i5,3f10,4i5,** with the following information expected

Columns	Variable	Description
1- 5	IELNUM	Element Number
6-10	IFACE	Number of face to which traction applied†
11-20	T1	Uniform x_1 traction magnitude
21-30	T2	Uniform x_2 traction magnitude
31-40	T3	Uniform x_3 traction magnitude
41-45	IGEN	Data Generation Parameter
		= 0: Do not use generation sequence
		= 1: Use a linear generation sequence
		= 2: Use a bilinear generation sequence
46-50	LCT1	Load Time Function Governing Traction 1 Component
51-55	LCT2	Load Time Function Governing Traction 2 Component
56-60	LCT3	Load Time Function Governing Traction 3 Component

For hexahedral and pentahedral elements, the IFACE definitions are as follows:

Hexahedral elements	
IFACE = 1	Face connecting nodes 1-2-3-4
IFACE = 2	Face connecting nodes 5-6-7-8
IFACE = 3	Face connecting nodes 1-5-6-2
IFACE = 4	Face connecting nodes 3-4-8-7
IFACE = 5	Face connecting nodes 2-3-7-6
IFACE = 6	Face connecting nodes 1-4-8-5
Pentahedral elements	
IFACE = 1	Face connecting nodes 1-2-3-4
IFACE = 2	Face connecting nodes 5-6-7-8
IFACE = 3	Face connecting nodes 1-5-6-2
IFACE = 4	Face connecting nodes 2-3-7-6
IFACE = 5	Face connecting nodes 1-3-7-5
IFACE = 6	Does not exist

If IGEN = 1 or 2, then FENDAC reads the next data line as telling how to generate surface traction data. The format of such lines is **4i5** with the following descriptions:

Columns	Variable	Description
1- 5	NINC(1)	Number of element increments in direction 1
6-10	INC(1)	Increment of element number in direction 1
11-15	NINC(2)	Number of element increments in direction 2
16-20	INC(2)	Increment of element number in direction 2

Terminate input of element surface traction data with a blank line.

16.3.7 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as follows:

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired:
		= 1: σ_{11}
		= 2: σ_{22}
		= 3: σ_{33}
		= 4: σ_{12}
		= 5: σ_{23}
		= 6: σ_{31}
		= 7: $\frac{1}{3}\mathbf{tr}(\boldsymbol{\sigma})$
		= 8: $\sqrt{J_2'}$
		= 9: $\|\|\|\mathbf{s}-\mathbf{q}\|\|\|$
		=10: ϵ_{11}
		=11: ϵ_{22}
		=12: ϵ_{33}
		=13: γ_{12}
		=14: γ_{23}
		=15: γ_{31}
		=16: $\mathbf{tr}\ (\boldsymbol{\epsilon})$
		=17: $\overline{e}^p$

Terminate input of element time history data with a blank line.

16.4 3-D BILINEAR DEGENERATED CONTINUUM SHELL ELEMENT

This element may be used in either quadrilateral (4-node) or trilateral (3-node) modes to solve 3-D forms of the momentum balance equation on shell domains:

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{g} = \rho \mathbf{a},$$

in which $\boldsymbol{\sigma}(\epsilon)$ is given by any one of a number of different stress-strain constitutive models. In the preceding expression, $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor, $\mathbf{g}$ denotes a gravitational body force vector, ρ the material mass density, and $\mathbf{a}$ the particle acceleration vector. This element can be used to solve either quasi-static problems (elliptic BVPs) or dynamic problems (hyperbolic BVPs). The introduction of shell geometrical and kinematic assumptions are discussed briefly below; they are discussed in more detail in Reference 2.

16.4.1 Shell Geometrical and Kinematic Assumptions

The geometry of the shell domain is described in terms of a parameterized reference surface $\overline{\mathbf{X}}(r,s) \in \Re^3$ and a corresponding vector thickness function $\hat{\mathbf{X}}(r,s,t) \in \Re^3$. The in-surface parametric natural coordinates of the shell are denoted by r and s, while the out-of-surface parametric coordinate is denoted by t. If the shell domain is discretized into a mesh of finite elements, the geometry for a given element which is typical in terms of the reference surface $\bar{\mathbf{X}}$ and the thickness vector $\hat{\mathbf{X}}$ can be expressed as follows:

$$\mathbf{X}(\xi,\eta,\zeta) = \overline{\mathbf{X}}(\xi,\eta) + \hat{\mathbf{X}}(\xi,\eta,\zeta) \tag{16.4.1}$$

in which $\xi,\eta,\zeta \in [-1,1]$ form the natural coordinate domain of the element which is the biunit cube [Figure 16.4]. The reference surface for a given element is written

$$\overline{\mathbf{X}}(\xi,\eta) = \sum_{a=1}^{nen} N_a(\xi,\eta)\overline{\mathbf{X}}_a \tag{16.4.2}$$

in which N_a are suitable two-dimensional shape functions and

$$\overline{\mathbf{X}}_a = \frac{1}{2}(1-\overline{\zeta})\mathbf{X}_a^- + \frac{1}{2}(1+\overline{\zeta})\mathbf{X}_a^+. \tag{16.4.3}$$

In the interest of handling shells of variable thickness, each node of the element possesses two coordinate points: $\mathbf{X}_a^+$ is the nodal location on the top surface ($\zeta = 1$) of the element and $\mathbf{X}_a^-$ is the position of the node on the bottom surface ($\zeta = -1$). The reference surface is fixed by selecting a value of $\overline{\zeta}$ on the interval $[-1,1]$ [Figure 16.5]. The significance of the reference surface becomes apparent when kinematics are discussed. Its selection is typically important only in the context of shell-continuum modeling. The thickness vector function for the element takes the form:

$$\hat{\mathbf{X}}(\xi,\eta,\zeta) = \sum_{a=1}^{nen} N_a(\xi,\eta) z_a(\zeta)\hat{\mathbf{F}}_a \tag{16.4.4}$$

where $\hat{\mathbf{F}}_a$ is a unit vector in the nodal fiber direction

$$\hat{\mathbf{F}}_a = \frac{\mathbf{X}_a^+ - \mathbf{X}_a^-}{\|\mathbf{X}_a^+ - \mathbf{X}_a^-\|} \tag{16.4.5}$$

in which $\|\cdot\|$ denotes the Euclidean norm of a vector. In the present formulation, the fiber direction is generally not orthogonal to the reference surface surface of the shell. The thickness function along the fiber direction is represented by $z_a(\zeta)$ which is written

$$z_a(\zeta) = \frac{1}{2}(1+\zeta)z_a^+ + \frac{1}{2}(1-\zeta)z_a^- \tag{16.4.6}$$

$$z_a^+ = \frac{1}{2}(1-\bar{\zeta})\|\mathbf{X}_a^+ - \mathbf{X}_a^-\| \tag{16.4.7a}$$

$$z_a^- = -\frac{1}{2}(1+\bar{\zeta})\|\mathbf{X}_a^- - \mathbf{X}_a^-\|. \tag{16.4.7b}$$

<u>**NOTE:**</u> For analysis with degenerated shell elements $2 * nsd$ coordinates are associated with each node. FENDAC expects $2 * nsd$ coordinates per node when the global control variable *ISHELL* is set equal to 1.

16.4.2 The Nodal Fiber Basis

To facilitate shell kinematics, orthonormal fiber bases are erected at each shell element node, and the rotational degrees of freedom at each node are computed with respect to the first two legs of the basis. The fiber direction is invariably in the direction $\mathbf{e}_3^f = \hat{\mathbf{F}}_a$ which need not be orthogonal to the shell's reference or lamina surface. The remaining two legs of a given nodal fiber basis $\mathbf{e}_1^f$ and $\mathbf{e}_2^f$ are chosen to be as closely aligned as is possible with the global $\mathbf{e}_1^g$ and the $\mathbf{e}_2^g$ vectors. The fiber basis generation algorithm at a given nodal point is as given in Box 16.4.1.

Let $a_i = |\hat{X}_i|$ **for** $i = 1,2,3$
$j = 1$
If $a_1 > a_3$ **then** $a_3 = a_1$, $j = 2$
If $a_2 > a_3$ **then** $j = 3$
$\mathbf{e}_3^f = \hat{\mathbf{X}}$
$\mathbf{e}_2^f = \frac{\hat{\mathbf{X}}\times\mathbf{e}_j}{\|\hat{\mathbf{X}}\times\mathbf{e}_j\|}$
$\mathbf{e}_1^f = \mathbf{e}_2^f \times \hat{\mathbf{X}}$

Box 16.4.1: Algorithm for computing fiber basis.

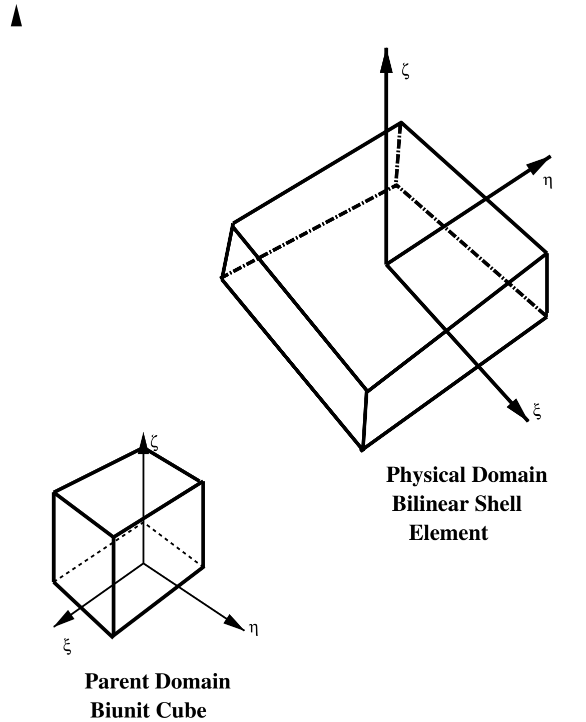


Figure 16.4: Parent and physical domains for 3-D bilinear degenerated shell element.

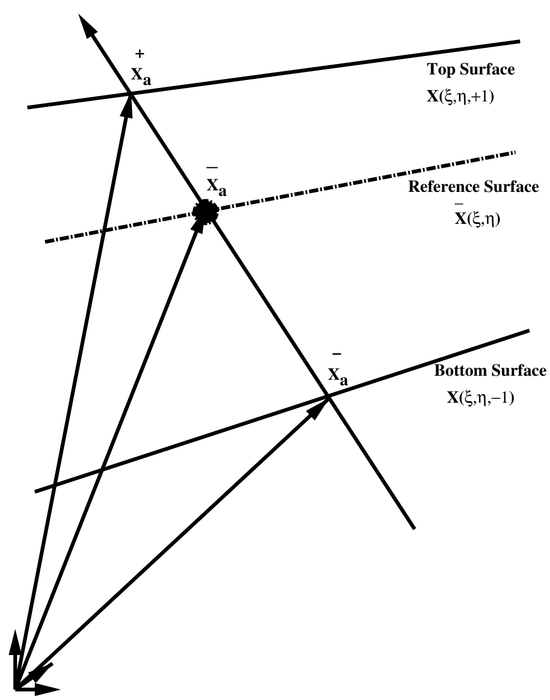


Figure 16.5: Upper and lower shell nodal coordinates and nodal fiber direction.

16.4.3 Shell Kinematics

Each shell element node possesses three translational degrees of freedom at the level of the reference surface ($\bar{u}_i^a$ for $i = 1, 2, 3$) and two rotational degrees of freedom (θ_j^a for $j = 1, 2$) taken about the first and second legs of the fiber basis $\mathbf{e}_1^f$ and $\mathbf{e}_2^f$, respectively. The infinitesimal displacement field throughout a given shell element, consistent with the corresponding assumption of infinitesimal rotations can be expressed with respect to the global basis in the following way:

$$\mathbf{u}(\xi, \eta, \zeta) = \bar{\mathbf{u}}(\xi, \eta) + \mathbf{U}(\xi, \eta, \zeta) \tag{16.4.8}$$

in which

$$\bar{\mathbf{u}}(\xi, \eta) = \sum_{a=1}^{nen} N_a(\xi, \eta)\bar{\mathbf{u}}_a \tag{16.4.9}$$

$$\mathbf{U}(\xi, \eta, \zeta) = \sum_{a=1}^{nen} N_a(\xi, \eta)\mathbf{U}_a(\zeta) \tag{16.4.10}$$

$$\mathbf{U}_a(\zeta) = z_a(\zeta)\hat{\mathbf{U}}_a \tag{16.4.11}$$

$$\hat{\mathbf{U}}_a = \theta_2^a \mathbf{e}_{a1}^f - \theta_1^a \mathbf{e}_{a2}^f \tag{16.4.12}$$

where: θ_1^a and θ_2^a are the infinitesimal rotations of the fiber at node a about the fiber basis vectors $\mathbf{e}_{a1}^f$ and $\mathbf{e}_{a2}^f$, respectively.

> **NOTE:** The employment of nodal fiber bases allows nodal restraints to be applied to elements in a fashion consistent with the element geometry. While this feature gives the analyst complete control over the application over rotational nodal restraints, it often requires some extra effort to prepare working data sets. To facilitate this procedure, it is often desirable to print the fiber basis at each shell node during the data checking process. The nodal fiber basis at each shell node will be printed if the element group input variable *ifchk* is set to unity.

Degenerated shell elements are capable of easily interfacing with continua when the continuum elements adjoin either the top of bottom surface of the shell. When, for example, the continuum rests atop the shell, then the reference surface is selected with $\zeta = 1$ as the top surface of the shell. Similarly, when the shell rests atop the continuum, the bottom surface becomes the reference surface by selecting $\zeta = -1$. These selections ensure C^0 continuity of displacements between the shell and continuum elements. For other types of shell-continuum interfacing, modified transition elements are typically required to obtain full compatibility.

16.4.4 INPUT OF SHELL ELEMENT DATA

Input for this element is comprised of up to six segments:

1. A master control line,
2. Input of material properties,
3. Element gravity vectors
4. Input of the reference surface parameter $\bar{\zeta}$.
5. Input of element connectivities,
6. Applied surface tractions (if used), and
7. Element time histories.

16.4.5 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 5
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)
16-20	MTYPE	Material Type
21-25	NSURF	Number of surface tractions to be applied
26-30	NEDGE	Number of edge tractions to be applied
31-35	IBBAR	Strain-Displacement Option
		IBBAR=0, Standard Formulation, full integration
		IBBAR=1, Reduced int. of transverse shear
		IBBAR=2, Uniformly reduced int. of membrane effects
		IBBAR=3, Selective reduced int. of membrane effects
		IBBAR=4, Selective reduced int. of membrane & transverse effects
		IBBAR=5, Reduced int. of membrane shear effects
36-40	NHIST	Number of element time histories desired
41-45	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
		= 2: Printout at element quadrature points
46-50	LCASG	Load-time function modulating element gravity
51-55	LCASP	Load-time function modulating X_1 surface tractions
56-60	LCASS	Load-time function modulating X_2 surface tractions
61-65	LCASE	Load-time function modulating X_3 surface tractions
66-70	IFCHK	Fiber basis print/generate restraint code option
		=-7 Restrain **all** shell node rotations
		=-6 Restrain shell node rotations about X_2 and X_3 axes
		=-5 Restrain shell node rotations about X_1 and X_3 axes
		=-4 Restrain shell node rotations about X_3 axis
		=-3 Restrain shell node rotations about X_1 and X_2 axes
		=-2 Restrain shell node rotations about X_2 axis
		=-1 Restrain shell node rotations about X_1 axis
		= 0 Omit printout of nodal fiber bases.
		= 1 Printout the nodal fiber bases.

(Continued Master Control Line Data . . .)

Columns	Variable	Description
71-75	NFINT	Number of fiber integration points to use through shell thickness
76-80	ITAN	Continuum/Consistent Tangent Option
		ITAN=0, use continuum tangent
		ITAN=1, use consistent tangent
81-85	IFRAC	Number of material volume fractions per element
		Used only for Topology Optimization Apps.
86-90	IMIX	Volume Fraction Mixing Rule on
		IMIX=0, Voigt mixing rule (constant strain)
		IMIX=1, Reuss mixing rule (constant stress)

16.4.6 Material Property Data

The material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one line for each material type. The format for each line of material property data set depends upon the type of three-dimensional constitutive model being used (MTYPE). Due to the shell stress assumption, additional restrictions are placed upon constitutive models. The current material models that can be employed with this shell element are:

Model 1 Linear isotropic/anisotropic elasticity

Model 2 Linearly hardening $J-2$ elastoplasticity models.

Model 9 Nonlinearly hardening $J-2$ elastoplasticity models.

Model 10 Hill's Orthotropic Elastoplasticity model with hardening.

Consult Section 17 for input formats for these material models.

16.4.7 Element Gravity Vectors

Each group of shell elements must have a gravity vector. The input format for this element type is **3f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity
21-30	Grav(3)	X_3 component of gravity

16.4.8 Element Reference Surface Parameter $\bar{\zeta}$

For each group of shell elements, the reference surface parameter $\bar{\zeta}$ must be specified. The input format for this parameter is **f10:**

Columns	Variable	Description
1-10	η	Reference surface parameter $-1 \leq \bar{\zeta} \leq 1$

16.4.9 Element Connectivity Data

Input of element connectivity for this element type is identical to that for the bilinear Laplacian and continuum elements described in Section 16.1.3. **Terminate input of element connectivity data with a blank line.**

16.4.10 Element Surface Traction Data

The format for input of element surface traction data is **2i5,3f10**. Currently no generation procedures are available to facilitate input of this data. Thus FENDAC expects NSURF lines of data with the following information expected on each line:

Columns	Variable	Description
1- 5	IELNUM	Element Number
6-10	IFACE	Number of face to which traction applied†
		1 for top face
		-1 for bottom face
11-20	PRESS	Uniform compressive normal stress
21-30	SHEAR1	Uniform shear stress in lamina $\mathbf{e}_1^l$ direction
31-40	SHEAR2	Uniform shear stress in lamina $\mathbf{e}_2^l$ direction

Terminate input of element surface traction data with a blank line.

16.4.11 Element Edge Load Data

The format for input of element edge load data is **3i5,2f10**. Currently no generation procedures are available to facilitate input of this data. Thus FENDAC expects NEDGE lines of data with the following information expected on each line:

Columns	Variable	Description
1- 5	IELNUM	Element Number
6-10	IEDGE	Element edge number
		connecting nodes 1-2: IEDGE=1
		connecting nodes 2-3: IEDGE=2
		connecting nodes 3-4: IEDGE=3
		connecting nodes 4-1: IEDGE=4
11-15	IEDOF	Associated DOF of applied traction
		=1: X_1 traction
		=2: X_2 traction
		=3: X_3 traction
		=4: Moment about $\mathbf{e}_1^f$
		=5: Moment about $\mathbf{e}_2^f$
16-25	EDGEF1	Traction magnitude at 1^{st} node
26-35	EDGEF2	Traction magnitude at 2^{nd} node

Terminate input of element edge load data with a blank line.

16.4.12 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as listed below. Each of these quantities are element volume averaged, and thus are strictly membrane quantities.

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired:
		= 1: σ_{11}
		= 2: σ_{22}
		= 3: σ_{33}
		= 4: σ_{12}
		= 5: σ_{23}
		= 6: σ_{31}
		= 7: $\frac{1}{3}\mathbf{tr}(\boldsymbol{\sigma})$
		= 8: $\sqrt{J_2'}$
		= 9: $\|\|\|\mathbf{s} - \mathbf{q}\|\|\|$
		=10: ϵ_{11}
		=11: ϵ_{22}
		=12: ϵ_{33}
		=13: γ_{12}
		=14: γ_{23}
		=15: γ_{31}
		=16: $\mathbf{tr}\,(\boldsymbol{\epsilon})$
		=17: $\overline{e}^{p}$

Terminate input of element time history data with a blank line.

16.5 2-D COMPATIBLE SURFACE ELEMENTS

These elements are used in stress-controlled homogenization of two-dimensional periodic composite solids. To obtain the "effective" stress-strain relations of a given composite, numerical property tests can be performed on a FEM model of the composite's unit cell. The essential function of compatible surface elements is to insure that the unit cell deforms in a "compatible" fashion during the property testing programme. Further details on this subject are provided in references 1 and 2.

A group of compatible surface elements comprises a surface pair on the boundary of the unit cell, which in 2-dimensions is a parallelogram, and in 3-dimensions is a parallelepiped. In general, for 2-d homogenization computations, two element groups of compatible surface elements are required, one for each surface pair; each element of a group consists of a pair of two-node linear sub-elements. For general 3-d homogenization, three element groups are usually required, and the elements of each group consist of pairs of 4-node bilinear surface elements.

Input of data for a given compatible 2-D surface element group consists of the following three segments:

1. A master control line,
2. A surface penalty parameter, and
4. Element connectivity data.

16.5.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 11
6-10	NUMEL	Number of elements in this group (> 0)
11-15	IPRNT	Print code for incompatible displacements
		=0, omit printout
		=1, include printout
16-20	NNODES	Number of nodes comprising each surface
21-25	MAXEL	Maximum number of CS elements of this group
		to which a node can belong.

16.5.2 Penalty Parameter

For each group, there is a single penalty parameter K. The larger K is specified, the more strictly the compatibility constraint is satisfied. The penalty parameter K follows immediately after the master control line in a free format data line.

16.5.3 Element Connectivity Data

Element connectivity data commences on the line immediately following the penalty parameter line. A 2-D compatible surface element consists of two corresponding line sub-elements. The first line element is defined by nodes N1 and N2, while the second line element is specified by nodes N3 and N4. Thus, each element is comprised of 4 nodes. Accordingly, input of element connectivity for this element type is identical to that for the 2-D Laplacian element described above. **Terminate input of element connectivity data for this element group with a blank line.**

16.6 3-D COMPATIBLE SURFACE ELEMENTS

These elements are used in stress-controlled homogenization of three-dimensional periodic composite solids. To obtain the "effective" stress-strain relations of a given composite, numerical property tests can be performed on a FEM model of the composite's unit cell. The essential function of compatible surface elements is to insure that the unit cell deforms in a "compatible" fashion the property testing. Further details on this subject are provided in references 1 and 2.

A group of compatible surface elements comprises a surface pair on the boundary of the unit cell, which in 3-dimensions is a parallelepiped. For general 3-d homogenization, three element groups are usually required to model the three surface pairs of a parallelepiped. The elements of each group consist of pairs of 4-node bilinear surface elements.

Input of data for a given compatible 3-D surface element group consists of the following three segments:

1. A master control line,
2. A surface penalty parameter, and
4. Element connectivity data.

16.6.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 12
6-10	NUMEL	Number of elements in this group (> 0)
11-15	IPRNT	Print code for incompatible displacements =0, omit printout =1, include printout
16-20	NNODES	Number of nodes comprising each surface
21-25	MAXEL	Maximum number of CS elements of this group to which a node can belong.

16.6.2 Penalty Parameter

For each group, there is a single penalty parameter K. The larger K is specified, the more strictly the compatibility constraint is satisfied. The penalty parameter K follows immediately after the master control line in a free format data line.

16.6.3 Element Connectivity Data

Element connectivity data commences on the line immediately following the penalty parameter line. A 3-D compatible surface element consists of two corresponding 4-node bilinear surface elements. The first sub-element is defined by nodes N1, N2, N3, and N4 (much like the degenerated continuum shell elements of Section 16.4) while the second sub-element is specified by corresponding nodes N5, N6, N7 and N8. Thus, each element is comprised of 8 nodes. (Triangular sub-elements can also be employed by repeating both nodes N3 and N4 and nodes N7 and N8.) The input of element connectivity for this element group is virtually identical to that for the 8 node hexahedral continuum element described in Section 16.3.4. **Terminate input of element connectivity data for this element group with a blank line.**

16.7 3-D BILINEAR REISSNER-MINDLIN PLATE-SHELL ELEMENT

This element may be used in either quadrilateral (4-node) or triangular (3-node) mode to solve the 3-D forms of the momentum balance equation:

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{g} = \rho \mathbf{a},$$

in which $\boldsymbol{\sigma}(\epsilon)$ is given by isotropic elasticity. In the preceding expression, $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor, $\mathbf{g}$ denotes a gravitational body force vector, ρ the material mass density, and $\mathbf{a}$ the particle acceleration vector. This element can be used to solve either quasi-static problems (elliptic BVPs) or dynamic problems (hyperbolic BVPs).

Input for this element is comprised of up to five segments:

1. A master control line,
2. Input of material properties,
3. Element gravity vectors,
4. Input of element connectivities, and
5. Element time histories.

16.7.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 13
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)
16-20	IBBAR	INTEGRATION OPTION
		IBBAR=0, Standard Formulation
		IBBAR=1, Reduced int. of transverse shear
		IBBAR=2, Reduced int. of bending
		IBBAR=3, Reduced int. of transverse shear
		IBBAR=4, Reduced int. of membrane
		IBBAR=5, Reduced int. of mem. shear & trans. shear
		IBBAR=6, Reduced int. of membrane & trans. shear
21-25	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
26-30	LCSSP	Load-time function modulating element normal tractions
31-35	LCASG	Load-time function modulating element gravity
36-40	IFRAC	Number of volume fractions per element
41-45	IMIX	Volume Fraction Mixing Code
		IMIX = 0, Constant strain mixing rule
		IMIX = 1, Constant stress mixing rule
		IMIX = 2, Mixed mixing rule
46-50	NHIST	Number of element time histories for this group.

16.7.2 Material Property Data

The material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one line for each material type. This element possesses only a linear elastic constitutive model for structural applications. Within each specification of an element group, there will generally be multiple materials defined. Each material must be defined constants such as Young's Modulus, Poisson's Ratio, Mass density, and element thickness. The input of material parameters follows the Master Control Line for an element group and precedes the Element Gravity Vector.

The format of the material parameter specification for each material model follows the format:

(a) Material Number (format: i5).

(b) Input of the **eoscon** vector (format: 25f10). The **eoscon** vector contains an assortment of material constants for the linear elastic model as specified below.

eoscon(1)=ρ (material mass density).

eoscon(2)=Young's modulus, **E**.

eoscon(3)=Poisson's ratio, ν.

eoscon(4)=Thickness, th.

eoscon(5)=Void-solid factor for topology design applications(default is 1.0e-06).

eoscon(6)-eoscon(25) are unused constants.

16.7.3 Element Gravity Vectors

Each group of elements must have a gravity vector. The input format for this element type is **2f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity
21-30	Grav(3)	X_3 component of gravity

16.7.4 Element Connectivity Data

Input of element connectivity for this element type is identical to that of the 2-D Laplacian element described in Section 16.1.3. **Terminate input of element connectivity data with a blank line.**

16.7.5 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as follows:

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired:
		= 1: σ_{11}
		= 2: σ_{22}
		= 3: σ_{12}
		= 4: σ_{33}
		= 5: $\frac{1}{3}\mathbf{tr}(\boldsymbol{\sigma})$
		= 6: $\sqrt{J_2'}$
		= 7: unused
		= 8: ϵ_{11}
		= 9: ϵ_{22}
		=10: γ_{12}
		=11: ϵ_{33}
		=12: $\mathbf{tr}\ (\epsilon)$
		=13: unsued

Terminate input of element time history data with a blank line.

16.8 2-D BILINEAR POROUS CONTINUUM ELEMENT

This element may be used in either quadrilateral (4-node) or triangular (3-node) mode to solve the planar or cylindrical forms of the coupled momentum balance equations:

$$\nabla \cdot \boldsymbol{\sigma}'^{s} - n^{s}\nabla p_{w} - \xi \cdot (\mathbf{v}^{s} - \mathbf{v}^{w}) + \rho^{s}\mathbf{b} = \rho^{s}\mathbf{a}^{s},$$

$$\rho^{w}(\mathbf{v}^{s} - \mathbf{v}^{w}) \cdot \nabla\mathbf{v}^{w} - n^{w}\nabla p_{w} + \xi \cdot (\mathbf{v}^{s} - \mathbf{v}^{w}) + \rho^{w}\mathbf{b} = \rho^{w}\mathbf{a}^{w},$$

in which $\boldsymbol{\sigma}(\boldsymbol{\epsilon})$ is given by any one of a number of different stress-strain constitutive models. In the preceding expression, $\boldsymbol{\sigma}'^{s}$ denotes the effective Cauchy stress tensor of solid phase, p_w denotes pore-fluid pressure, n^{α} (where α can be s or w) denotes volume fraction of each phase, $\mathbf{v}^{\alpha}$, denotes velocity vector of each phase, $\mathbf{a}^{\alpha}$ denotes acceleration vector of each phase, ρ^{α} is the material mass density, and $\mathbf{b}$ denotes a gravitational body force vector. This element can be used to solve either quasi-static problems (parabolic BVPs) or dynamic problems (hyperbolic BVPs).

Input for this element is comprised of up to six segments:

1. A master control line,
2. Input of material properties,
3. Element gravity vectors
4. Input of element connectivities,
5. Applied surface tractions (if used), and
6. Element time histories.

16.8.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 14
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(1 or 2)
16-20	MTYPE	Material Type
21-25	IOPT	Analysis Option
		IOPT=0, plane strain analysis
		IOPT=1, plane stress analysis
		IOPT=2, cylindrically symmetric
26-30	IMIX	Volume Fraction Mixing Code
		IMIX = 0, Constant strain mixing rule
		IMIX = 1, Constant stress mixing rule
31-35	NSURF	Number of surface tractions to be applied
		Surface Traction Follows Connectivity Data
36-40	IBBAR	Strain-Displacement Option
		IBBAR=0, Standard Formulation
		IBBAR=1, Mean-Dilatational Formulation
41-45	NHIST	Number of element time histories desired
		Element time history data follows element
		connectivity and surface traction data
46-50	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
		= 2: Printout at element quadrature points
51-55	LCASG	Load-time function modulating element gravity
56-60	LCASP	Load-time function modulating element normal tractions
61-65	LCASS	Load-time function modulating element shear tractions
66-70	ITAN	Continuum/Consistent Tangent Option
		= 0: Continuum Tangent Operators
		= 1: Consistent Tangent Operators
71-75	NQUAD	Number of quadrature points per element (1 or 4)
76-80	IFRAC	Number of volume fractions per element

16.8.2 Material Property Data

The material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one line for each material type. The format for solid phase line of material property data set depends upon the type of constitutive model being used. (Consult Section 17 for specific models).

The format for fluid phase line of material property data is **4F10**. The material properties to be entered are of the form:

Columns	Variable	Description
1-10	ρ	Fluid density
11-20	n^w	Volume fraction of fluid(=porosity)
21-30	κ	Permeability
31-40	λ^w	Fluid bulk modulus

16.8.3 Element Gravity Vectors

Each group of continuum elements must have a gravity vector. The input format for this element type is **2f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity

16.8.4 Element Connectivity Data

Input of element connectivity for this element type is identical to that of the 2-D Laplacian element described in Section 16.1.3. **Terminate input of element connectivity data with a blank line.**

16.8.5 Element Surface Traction Data

Both pressure and shear tractions can be applied to the edges of quadrilateral continuum elements. Sign conventions for positive pressure tractions and positive shear tractions are shown in Figures 16.2a and 16.2b, respectively. The applied tractions can vary linearly along the sides as shown. The traction forces are applied for solid and fluid separately.

The format for input of element surface traction data is **2i5,5f10,i5,** with the following information expected

Columns	Variable	Description
1- 5	IELNUM	Element Number
6-10	NSIDE	Number of side to which traction applied
		connecting nodes 1-2: NSIDE=1
		connecting nodes 2-3: NSIDE=2
		connecting nodes 3-4: NSIDE=3
		connecting nodes 4-1: NSIDE=4
11-20	P1	left node normal traction magnitude
21-30	P2	right node normal traction magnitude
31-40	S1	left node shear traction magnitude
41-50	S2	right node shear traction magnitude
51-60	Parameter	Parameter for each phase
		=0.0: apply force to fluid
		=1.0: apply force to solid
61-65	IGEN	Data Generation Parameter
		= 0: Do not use generation sequence
		= 1: Use a linear generation sequence

When IGEN = 1, then surface traction data is generated along a sequence of elements using the following additional data having the format **2i5:**

Columns	Variable	Description
1- 5	NINC	Number of element increments
6-10	INC	Increment of element number

Terminate input of element surface traction data with a blank line.

16.8.6 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as follows:

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired:
		= 1: σ_{11}
		= 2: σ_{22}
		= 3: σ_{12}
		= 4: σ_{33}
		= 5: $\frac{1}{3}\mathbf{tr}(\boldsymbol{\sigma})$
		= 6: $\sqrt{J_2'}$
		= 7: $\lVert\mathbf{s}-\mathbf{q}\rVert$
		= 8: ϵ_{11}
		= 9: ϵ_{22}
		=10: γ_{12}
		=11: ϵ_{33}
		=12: $\mathbf{tr}\ (\epsilon)$
		=13: $\overline{e}^p$
		=14: p_w

Terminate input of element time history data with a blank line.

16.9 3D TRILINEAR ELASTIC BIOT FLUID-SOLID CONTINUUM ELEMENT

This element may be used in either hexahedral (8-node), pentahedral (6-node), or tetrahedral (4-node) modes to solve 3-D elastic porous medium problems of the form:

$$\nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{g} = \rho \ddot{\mathbf{u}}$$

$$-\nabla p_f - \frac{1}{n}\mathbf{R} \cdot \dot{\mathbf{w}} + \rho_f \mathbf{g} = \rho_f(\ddot{\mathbf{u}} + \frac{1}{n}\ddot{\mathbf{w}})$$

in which:

$\boldsymbol{\sigma}$	is the total stress in the porous medium;
p_f	is the fluid pressure;
$\mathbf{R}$	is the resistivity tensor (inverse of permeability);
ρ_f	is the intrinsic fluid density;
ρ_s	is the intrinsic solid density;
ρ	is the bulk density of the medium;
n	is the porosity of the medium;
$\mathbf{g}$	is a body force per unit mass;
$\mathbf{u}$	is the in-phase displacement of the fluid-solid medium;
$\mathbf{w}$	is the out-of-phase fluid displacement relative to the solid;

This element can be used to solve either hyperbolic dynamic problems (equations shown above) or quasi-static parabolic problems where inertial terms in the above equations are neglected.

Input for this element is comprised of up to six segments:

1. A master control line (required),
2. Input of material properties (required),
3. Element gravity vectors (required),
4. Input of element connectivities (required),
5. Applied surface tractions, (optional) and
6. Element time histories (optional).

16.9.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 15
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)
16-20		Unused
21-25		Unused
26-30		Unused
31-35	NSURF	Number of surface tractions to be applied
		Surface Traction Data Follows Connectivity Data
36-40	IBBAR	Strain-Displacement Option
		IBBAR=0, Standard Formulation
		IBBAR=1, Mean-Dilatational Formulation
41-45	NHIST	Number of element time histories desired
		Element time history data follows element
		connectivity and surface traction data
46-50	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
		= 2: Printout at element quadrature points
51-55	LCASG	Load-time function modulating element gravity
56-60	LCAST1	Load-time function modulating X_1 surface tractions
61-65	LCAST2	Load-time function modulating X_2 surface tractions
66-70	LCAST3	Load-time function modulating X_3 surface tractions
71-75	NQUAD	Number of quadrature points per element (1 or 8)
76-80		Unused

16.9.2 Material Property Data

The material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one for each different material. The format for each material property data set is described below. Presently, this element features strictly linear elastic material behaviours of the following form:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{31} \\ p_f \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 & C_{17} \\ C_{21} & C_{22} & C_{23} & 0 & 0 & 0 & C_{27} \\ C_{31} & C_{32} & C_{33} & 0 & 0 & 0 & C_{37} \\ 0 & 0 & 0 & C_{44} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{55} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} & 0 \\ C_{71} & C_{72} & C_{73} & 0 & 0 & 0 & C_{77} \end{bmatrix} \cdot \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \\ \zeta \end{bmatrix}$$

In addition, fluid flow in the porous medium follows Darcy's law:

$$\begin{bmatrix} \dot{w}_1 \\ \dot{w}_2 \\ \dot{w}_3 \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} \cdot \begin{bmatrix} (p_f)_{,1} \\ (p_f)_{,2} \\ (p_f)_{,3} \end{bmatrix}$$

The material densities, and permeability are entered in the eoscon array in 25f10 format as follows:

Parameter	Constant	Description
eoscon(1)	ρ_s	Intrinsic solid mass density
eoscon(2)	ρ_f	Intrinsic fluid mass density
eoscon(3)	n	Porosity
eoscon(4)	K_f	Fluid bulk modulus
eoscon(5)	k_{11}	permeability
eoscon(6)	k_{22}	permeability
eoscon(7)	k_{33}	permeability
eoscon(8)	k_{23}	permeability
eoscon(9)	k_{13}	permeability
eoscon(10)	k_{12}	permeability

The elasticity coefficients are entered in 5f10 format in the following order:

$$\begin{aligned} &C_{11}, C_{22}, C_{33}, C_{44}, C_{55} \\ &C_{66}, C_{77}, C_{37}, C_{27}, C_{17} \\ &C_{23}, C_{13}, C_{12} \end{aligned}$$

16.9.3 Element Gravity Vectors

Each group of continuum elements must have a gravity vector. The input format for this element type is **3f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity
21-30	Grav(3)	X_3 component of gravity

16.9.4 Element Connectivity Data

Input of element connectivity for this element is precisely the same as that of the trilinear continuum element described in Section 16.3.4.

16.9.5 Element Surface Traction Data

The format for input of element surface traction data is **2i5,3f10,4i5,** is precisely the same as that for the trilinear continuum element described in Section 16.3.6.

16.9.6 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as follows:

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired:
		1: σ_{11}
		2: σ_{22}
		3: σ_{33}
		4: σ_{12}
		5: σ_{23}
		6: σ_{31}
		7: mean normal total stress, $\frac{1}{3}\mathbf{tr}\boldsymbol{\sigma}$
		8: $\sqrt{J_2'}$
		9: p_f
		10: ϵ_{11}
		11: ϵ_{22}
		12: ϵ_{33}
		13: γ_{12}
		14: γ_{23}
		15: γ_{31}
		16: **tr** (ϵ)
		17: ζ (change of fluid content)
		18: rate of dissipation for full element group
		19: total energy dissipated for this group

Terminate input of element time history data with a blank line.

16.10 2-D/3-D LINEAR ELASTIC TRUSS ELEMENTS

Linear elastic truss elements can be used in solving quasi-static structural boundary value problems. Input of data for a group of 2-D or 3-D truss elements consists of the following three segments:

1. A master control line,
2. Truss element physical parameters, and
3. Element connectivity data.
4. Ground structure topology optimization information.

16.10.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 1 for truss elements.
6-10	NUMEL	Number of truss elements in this group (> 0)
11-15	NUMAT	Number of different types of truss elements in this particular group. Each type have their own set of properties data
16-20	IDR	Displacement/Rotation Indicator Not presently used
21-25	NHIST	Number of element time histories Not presently used
26-30	IPRNT	Option for printing our element forces, etc. Eq. 0: Omit printout Eq. 1: Perform the printout
31-35	ICOMP	Compression Only Option Not presently used
36-40	IDES	Ground Structure Topology Optimization Eq. 0: Optimization not performed Eq. 1: Optimization performed Requires additional input (see below)
41-45	IMAST	Symmetry Control with Topology Optimization Eq. 0: Not used Eq. 1: Used (requires DESVAR_MAP file)

16.10.2 Truss Element Physical Properties

For each type of element within the overall group, it is necessary to specify the cross-sectional area, Young's modulus, moment of inertia, and mass density. A single line of format-free data is required for each type of truss elements . Each line of data must include the following values:

1. The material group number;
2. The cross-sectional area of the truss element in appropriate units;
3. The Young's modulus of the material comprising the truss element;
4. The minimum moment of inertia of the truss cross-section. (This information will be used to compute critical buckling loads for each truss element.)
5. The mass density of the material.

There should be exactly NUMAT lines of truss element physical properties data.

16.10.3 Element Connectivity Data

Element connectivity data commences on the line immediately following the truss element physical properties data. A truss element consists of a single line segment with area and stiffness properties that connects two nodal points. The linear truss elements are defined by: (1) their element number; (2) their physical properties type; and (3) the two nodes which they connect. Accordingly, input of element connectivity for the linear truss elements is as listed below:

Element Connectivity Data Line

Notes	Columns	Variable	Description
(1)	1- 5	N	Element number ($0 < N \leq NUMEL$)
	6-10	MAT(N)	Material set number (≥ 0)
	11-15	IEN(1,N)	Number of 1^{st} node
	16-20	IEN(2,N)	Number of 2^{nd} node
(4)	51-55	NG	Generation parameter
			if 0, no generation
			if ≥ 1, generate element data

Connectivity data for truss elements can be input either with a single line of data for each element (NG=0), or, for certain classes of problems, using automatic data generation (NG>0). See Figure 16.1 for a schematic representation of the generation scheme. The input format for each line specifying the generation scheme is **6I5**, and the meaning of the data is as described below.

Columns	Variable	Description
1- 5	NEL(1)	Number of elements in direction 1 $\geq$ 0; if = 0, set internally to 1
6-10	INCEL(1)	Element number increment for direction 1 if = 0, set internally to 1
11-15	INC(1)	Node number increment for direction 1 if = 0, set internally to 1
16-20	NEL(2)	Number of elements in direction 2 $\geq$ 0; if = 0, set internally to 1
21-25	INCEL(2)	Element number increment for direction 2 if = 0, set internally to 1
26-30	INC(2)	Node number increment for direction 2 if = 0, set internally to 1

16.10.4 Topology Optimization Data

To define the topology optimization problem, the following design information is required and can be entered in unformatted form:

Maximum mass of the structure
Maximum compliance of the structure under specified loading
Allowable stress
Minimum factor of safety against buckling
Maximum number of elements to keep in the optimal structure.

Terminate input of element for this group with a blank line.

16.11 2-D BIQUADRATIC TRIANGULAR CONTINUUM ELEMENT

This 6-noded element can be used to solve the planar or cylindrical forms of the momentum balance equation:

$$\nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{g} = \rho \mathbf{a},$$

in which $\boldsymbol{\sigma}(\epsilon)$ is given by any one of a number of different stress-strain constitutive models. In the preceding expression, $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor, $\mathbf{g}$ denotes a gravitational body force vector, ρ the material mass density, and $\mathbf{a}$ the particle acceleration vector. This element can be used to solve either quasi-static problems (elliptic BVPs), parbolic BVPs, or dynamic problems (hyperbolic IBVPs).

Input for this element is comprised of up to five segments:

1. A master control line,
2. Input of material properties,
3. Element gravity vectors
4. Input of element connectivities, and
5. Element time histories.

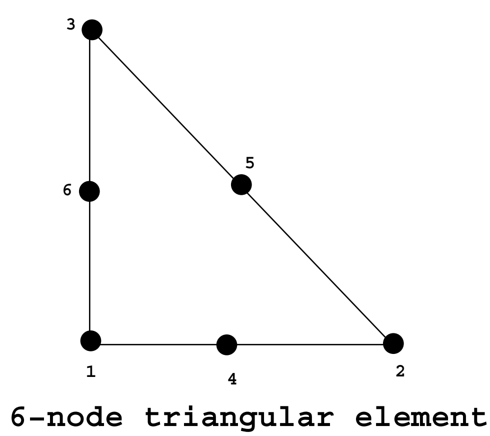


Figure 16.11: Six-noded biquadratic triangular continuum element.

16.11.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 16
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)
16-20	MTYPE	Material Type
21-25	IOPT	Analysis Option
		IOPT=0, plane strain analysis
		IOPT=1, plane stress analysis
		IOPT=2, cylindrically symmetric
26-30	IFD	Finite Deformation Option
		IFD = 0, Neglect finite deformations
		IFD = 1, Account for finite deformations
31-35	NSURF	Presently not used.
36-40	IBBAR	Strain-Displacement Option
		IBBAR=0, Standard Formulation
		IBBAR=1, Mean-Dilatational Formulation
41-45	NHIST	Number of element time histories desired
		Element time history data follows element
		connectivity and surface traction data
46-50	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
		= 2: Printout at element quadrature points
51-55	LCASG	Load-time function modulating element gravity
56-60	LCASP	Load-time function modulating element normal tractions
61-65	LCASS	Load-time function modulating element shear tractions
66-70	ITAN	Continuum/Consistent Tangent Option
		= 0: Continuum Tangent Operators
		= 1: Consistent Tangent Operators
71-75	NQUAD	Number of quadrature points per element (1 or 3)
76-80	IFRAC	Not used.

Columns	Variable	Description
81-85	ICHECK	Not used.
86-90	IMIX	Not used.
91-95	IMATIN	Code for special input format of material data
		= 0: Standard input format
		= 1: Input in a separate file MATERIAL.data

16.11.2 Material Property Data

When IMATIN=0 (standard material input mode), the material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one line for each material type. The format for each line of material property data set depends upon the type of constitutive model being used. Consult Section 17 for specific models.

On the other hand, when IMATIN=1, the mode for input of material data is different. With IMATIN=1, it is assumed that each element will have its own set of material properties. As this option is available only with linear elasticity, the input format for the file MATERIAL.data is as follows.

element number, n
mass density, rho
$C_{11}, C_{12}, C_{13}, C_{14}$
$C_{21}, C_{22}, C_{23}, C_{24}$
$C_{31}, C_{32}, C_{33}, C_{34}$
$C_{41}, C_{42}, C_{43}, C_{44}$

Each line of data is format free, but there must be NUMEL sets of data, with no blank lines between sets.

16.11.3 Element Gravity Vectors

Each group of continuum elements must have a gravity vector. The input format for this element type is **2f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity
21-30	PGLOB	Parameter used in hybrid Voigt-Reuss
		mixing rule with topology optimization

16.11.4 Element Connectivity Data

See Figure 16.11 for a schematic representation of the node-numbering scheme for this element. Note that the nodes should be ordered in a counterclockwise fashion as shown. The input format for each line of data defining an element is 9I5. Input of element connectivity for this element type is as follows: **Terminate input of element connectivity data with a blank line.**

Columns	Variable	Description
1- 5	N	Element number
6-10	MAT(N)	Material identification number
11-15	IEN(1,N)	Number of element's first node
16-20	IEN(2,N)	Number of element's second node
21-25	IEN(3,N)	Number of element's third node
26-30	IEN(4,N)	Number of element's fourth node
31-35	IEN(5,N)	Number of element's fifth node
36-40	IEN(6,N)	Number of element's sixth node
41-45	NG	Code for usage of element generation If 0, no generation If 1, generation used

16.11.5 Generation Data Input

Columns	Variable	Description
1- 5	NEL(1)	Number of elements in direction 1 $\geq$ 0; if = 0, set internally to 1
6-10	INCEL(1)	Element number increment for direction 1 if = 0, set internally to 1
11-15	INC(1)	Node number increment for direction 1 if = 0, set internally to 1
16-20	NEL(2)	Number of elements in direction 2 $\geq$ 0; if = 0, set internally to 1
21-25	INCEL(2)	Element number increment for direction 2 if = 0, set internally to 1
26-30	INC(2)	Node number increment for direction 2 if = 0, set internally to 1

16.11.6 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as follows:

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired:
		= 1: σ_{11}
		= 2: σ_{22}
		= 3: σ_{12}
		= 4: σ_{33}
		= 5: $\frac{1}{3}\mathbf{tr}(\boldsymbol{\sigma})$
		= 6: $\sqrt{J_2'}$
		= 7: $\|\|\|\mathbf{s} - \mathbf{q}\|\|\|$
		= 8: ϵ_{11}
		= 9: ϵ_{22}
		=10: γ_{12}
		=11: ϵ_{33}
		=12: $\mathbf{tr}\ (\epsilon)$
		=13: $\overline{e}^p$

Terminate input of element time history data with a blank line.

16.12 3-D TRIQUADRATIC TETRAHEDRAL CONTINUUM ELEMENT

This 10-noded element is used to solve 3-D forms of the momentum balance equation:

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{g} = \rho \mathbf{a},$$

in which $\boldsymbol{\sigma}(\epsilon)$ is given by any one of a number of different stress-strain constitutive models. In the preceding expression, $\boldsymbol{\sigma}$ denotes the Cauchy stress tensor, $\mathbf{g}$ denotes a gravitational body force vector, ρ the material mass density, and $\mathbf{a}$ the particle acceleration vector. This element can be used to solve either quasi-static problems (elliptic BVPs) or dynamic problems (hyperbolic BVPs).

Input for this element is comprised of up to five segments:

1. A master control line,
2. Input of material properties,
3. Element gravity vectors
4. Input of element connectivities, and
5. Element time histories.

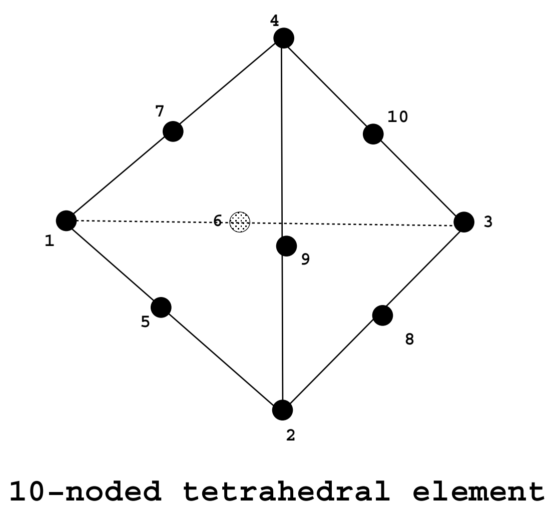


Figure 16.12: Schematic of recommended node numbering sequence for the 10-noded triquadratic tetrahedral continuum element.

16.12.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 17
6-10	NUMEL	Number of elements in this group (> 0)
11-15	NUMAT	Number of material types in this group(> 0)
16-20	MTYPE	Material Type
21-25	ITAN	Continuum/Consistent Tangent Option
		ITAN=0, use continuum tangent
		ITAN=1, use consistent tangent
26-30	IFD	Finite Deformation Code
		IFD = 0, Neglect Finite Deformation
		IFD $>$ 0, Include Finite Def. Effects
31-35	NSURF	Not used.
36-40	IBBAR	Strain-Displacement Option
		IBBAR=0, Standard Formulation
		IBBAR=1, Mean-Dilatational Formulation
41-45	NHIST	Number of element time histories desired
		Element time history data follows element
		connectivity and surface traction data
46-50	IPRINT	Stress-Strain Printout Option Code
		= 0: No printout of stresses & strains
		= 1: Printout at element centroid
		= 2: Printout at element quadrature points
51-55	LCASG	Load-time function modulating element gravity
56-60	LCAST1	Load-time function modulating X_1 surface tractions
61-65	LCAST2	Load-time function modulating X_2 surface tractions
66-70	LCAST3	Load-time function modulating X_3 surface tractions
71-75	NQUAD	Number of quadrature points per element (1 or 4)
76-80	IFRAC	Number of volume fractions per element (generally used
		in conjunction with topology optimization.)

16.12.2 Material Property Data

The material property data follows immediately after the master control line, and there must be NUMAT sets of material property data, one line for each material type. The format for each line of material property data set depends upon the type of constitutive model being used (MTYPE). Consult Section 17 for specific material models.

16.12.3 Element Gravity Vectors

Each group of continuum elements must have a gravity vector. The input format for this element type is **3f10.**

Columns	Variable	Description
1-10	Grav(1)	X_1 component of gravity
11-20	Grav(2)	X_2 component of gravity
21-30	Grav(3)	X_3 component of gravity

16.12.4 Element Connectivity Data

Ordering of nodes for this element should follow the example shown in Figure 16.12. Input of element connectivity for this element type can be used with or without data generation. Data for definition of a single element is expected in the format **13i5** in the following sequence:

Element Connectivity Data Line

Columns	Variable	Description
1- 5	N	Element number ($0 < N \leq NUMEL$)
6-10	MAT(N)	Material set number ($\geq$ 0)
11-15	IEN(1,N)	Number of 1^{st} node
16-20	IEN(2,N)	Number of 2^{nd} node
21-25	IEN(3,N)	Number of 3^{rd} node
26-30	IEN(4,N)	Number of 4^{th} node
31-35	IEN(5,N)	Number of 5^{th} node
36-40	IEN(6,N)	Number of 6^{th} node
41-45	IEN(7,N)	Number of 7^{th} node
46-50	IEN(8,N)	Number of 8^{th} node
51-55	IEN(9,N)	Number of 9^{th} node
56-60	IEN(10,N)	Number of 10^{th} node
61-65	NG	Generation parameter
		if 0, no generation
		if $\geq$ 1, generate element data

16.12.5 Element Data Generation Input

The input format such lines is **9i5.**

Columns	Variable	Description
1- 5	NEL(1)	Number of elements in direction 1 $\geq$ 0; if = 0, set internally to 1
6-10	INCEL(1)	Element number increment for direction 1 if = 0, set internally to 1
11-15	INC(1)	Node number increment for direction 1 if = 0, set internally to 1
16-20	NEL(2)	Number of elements in direction 2 $\geq$ 0; if = 0, set internally to 1
21-25	INCEL(2)	Element number increment for direction 2 if = 0, set internally to 1
26-30	INC(2)	Node number increment for direction 2 if = 0, set internally to 1
31-35	NEL(3)	Number of elements in direction 3 $\geq$ 0; if = 0, set internally to 1
36-40	INCEL(3)	Element number increment for direction 3 if = 0, set internally to 1
41-45	INC(3)	Node number increment for direction 3 if = 0, set internally to 1

16.12.6 Element Time History Data

For the input of element time history data, FENDAC expects NHIST lines of data to specify the requested element time histories. The format of each of the NHIST lines is **2i5**, with the data on each line as follows:

Columns	Variable	Description
1- 5	NEL	Number of element in which history is desired.
6-10	IQ	Quantity desired: = 1: σ_{11} = 2: σ_{22} = 3: σ_{33} = 4: σ_{12} = 5: σ_{23} = 6: σ_{31} = 7: $\frac{1}{3}\mathbf{tr}(\boldsymbol{\sigma})$ = 8: $\sqrt{J_2'}$ = 9: $\|\|\|\mathbf{s}-\mathbf{q}\|\|\|$ =10: ϵ_{11} =11: ϵ_{22} =12: ϵ_{33} =13: γ_{12} =14: γ_{23} =15: γ_{31} =16: $\mathbf{tr}\,(\boldsymbol{\epsilon})$ =17: $\overline{\epsilon}^p$

Terminate input of element time history data with a blank line.

16.13 2-D/3-D LINEAR BOUNDARY SPRING ELEMENTS

Linear boundary spring elements can be used in solving quasi-static structural boundary value problems. Input of data for a group of 2-D or 3-D boundary spring elements consists of the following three segments:

1. A master control line,
2. Spring element physical parameters, and
3. Element definitions.

16.13.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 9 for boundary springs.
6-10	NUMEL	Number of spring elements in this group (> 0)
11-15	NUMAT	Number of different types of spring elements in this particular group. Each type have their own set of properties data
16-20	IDR	Displacement/Rotation Indicator (Not presently used)
21-25	NHIST	Number of element time histories (Not presently used)
26-30	IPRNT	Option for printing our element forces, etc. Eq. 0: Omit printout Eq. 1: Perform the printout

16.13.2 Spring Element Physical Properties

For each type of element within the overall group, it is necessary to specify the direction in which the spring acts, and the stiffness of the spring in that direction. There should be NUMAT single lines of formatted data, and for each line, which represents a group of springs with the same properties, the following data are required:

Columns	Variable	Description
1- 5	M	An integer indicator ($1 \leq M \leq NUMAT$) for this group of springs.
11-20	DIR(1,M)	X1 Component of spring director
21-30	DIR(2,M)	X2 Component of spring director
31-40	DIR(3,M)	X3 Component of spring director
41-50	K(M)	Spring Stiffness
51-55	LCASM(M)	Load-time function used to modulate spring stiffness

Again, there should be exactly NUMAT lines of boundary spring element physical properties data.

16.13.3 Element Connectivity Data

Element connectivity data commences on the line immediately following the boundary spring element physical properties data. A boundary spring element consists of a single line segment with area and stiffness properties that connects two nodal points. The linear truss elements are defined by: (1) their element number; (2) their physical properties type; and (3) the two nodes which they connect. Accordingly, input of element connectivity for the linear truss elements is as listed below:

Element Connectivity Data Line

Columns	Variable	Description
1- 5	N	Element number ($0 < N \leq NUMEL$)
6-10	MAT(N)	Material set number (≥ 0)
11-15	IEN(N)	Number of the node restrained by the spring

16.14 2-D/3-D NODAL SPRING ELEMENTS

Nodal spring elements can be used in solving quasi-static structural boundary value problems. Input of data for a group of 2-D or 3-D nodal spring elements consists of the following three segments:

1. A master control line,
2. Spring element physical parameters, and
3. Element definitions.

16.14.1 Master Control Line

The format for this line is 20i5.

Columns	Variable	Description
1- 5	NTYPE	The number is 10 for nodal springs.
6-10	NUMEL	Number of spring elements in this group (> 0)
11-15	NUMAT	Number of different types of spring elements in this particular group. Each type have their own set of properties data
16-20	IDR	Displacement/Rotation Indicator (Not presently used)
21-25	NHIST	Number of element time histories (Not presently used)
26-30	IPRNT	Option for printing our element forces, etc. Eq. 0: Omit printout Eq. 1: Perform the printout
31-35	ICOMP	Option for compression-only spring behavior Eq. 0: Springs have tension/compression stiffness Eq. 1: Sprints have only compression stiffness

16.14.2 Spring Element Physical Properties

For each type of element within the overall group, it is necessary to specify the direction in which the spring acts, and the stiffness of the spring in that direction. There should be NUMAT single lines of formatted data, and for each line, which represents a group of springs with the same properties, the following data are required:

For NSD = 3:

Columns	Variable	Description
1- 5	M	An integer indicator ($1 \leq M \leq NUMAT$) for this group of springs.
11-20	DIR(1,M)	X1 Component of spring director
21-30	DIR(2,M)	X2 Component of spring director
31-40	DIR(3,M)	X3 Component of spring director
41-50	K_0(M)	Spring Stiffness
51-60	δ(M)	A relative spring deformation used with (ICOMP = 1)

For NSD = 2:

Columns	Variable	Description
1- 5	M	An integer indicator ($1 \leq M \leq NUMAT$) for this group of springs.
11-20	DIR(1,M)	X1 Component of spring director
21-30	DIR(2,M)	X2 Component of spring director
31-40	K_0(M)	Spring Stiffness
41-50	δ(M)	A relative spring deformation used with (ICOMP = 1)

16.14.3 Compression-Only Spring Behavior

The compression-only spring behavior can sometimes be used to achieve the effect of a contact surface. For example, consider two points A and B that are connected by a nodal spring element. If both points lie on surfaces, then the outward normal to the surface on which point A lies at point A is denoted $\mathbf{n}_A$, and the outward normal to the surface on which point B lies at point B is denoted $\mathbf{n}_B$ (Figure 16.14). Since it is assumed that $\mathbf{n}_A = -\mathbf{n}_B$, the normal at point A $\mathbf{n}_A$ is taken to be the element's director.

When the two points A and B have respective displacements $\mathbf{u}_A$ and $\mathbf{u}_B$ then the quantity $\Delta = \mathbf{u}_A \cdot \mathbf{n}_A + \mathbf{u}_B \cdot \mathbf{n}_B$ denotes the compression of the spring element connecting these two points. When $\Delta > 0$ the spring is in compression, and when $\Delta < 0$ the spring is in tension. To achieve continuous spring stiffness that vanishes with tension, the following equation is used to govern the spring stiffness when ICOMP = 1.

$$K(M) = \frac{K_0(M)}{2}\left[1 + erf\left(\frac{\Delta}{\delta(M)}\right)\right]$$

where

$$erf(x) = \frac{2}{\sqrt{\pi}}\int_0^x \exp(-u^2)\, du.$$

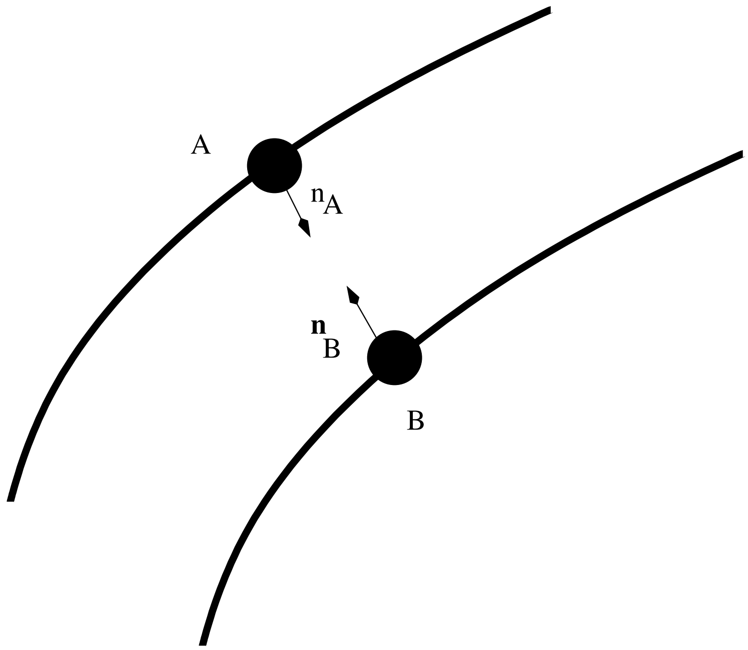


Figure 16.14: Schematic of nodal spring behaviors with (ICOMP = 1).

Again, there should be exactly NUMAT lines of boundary spring element physical properties data.

16.14.4 Element Connectivity Data

Element connectivity data commences on the line immediately following the nodal spring element physical properties data. A nodal spring element consists of a spring type (director, stiffness K_0) and the node numbers of the nodes it links together. Accordingly, input of element connectivity for the nodal spring elements is as listed below:

Element Connectivity Data Line

Columns	Variable	Description
1- 5	N	Element number ($0 < N \leq NUMEL$)
6-10	MAT(N)	Material set number (≥ 0)
11-15	IEN(1,N)	Number of first node
16-20	IEN(2,N)	Number of second node

17. MATERIAL MODEL'S DOCUMENTATION

FENDAC possesses a number of elasto-plastic constitutive models for continuum and structural applications. Within each specification of an element group, there will generally be multiple materials defined. Each material must have the same general constitutive equations (*i.e.* material model number), although specific parameters will be different for each material. The input of material parameters follows the **Master Control Line** for an element group and precedes the **Element Connectivity Data.**

The general format of material parameter specification for each material model follows the format:

(a) Material Number (format: **i5**).

(b) Input of the **eoscon** vector (format: **25f10**). The **eoscon** vector contains an assortment of material constants unique to each model.

(c) Elastic constants (see Section 16.1 for input format).

(d) Material directors (for anisotropic material models only).

(e) Plastic hardening tensors (for anisotropic plasticity models only).

A brief summary of each model and its associated material parameter input specifications are summarized in the following subsections. The currently available material model types are as listed below:

AVAILABLE MATERIAL MODEL TYPES

17.1 MODEL #1: LINEAR ANISOTROPIC ELASTICITY

17.1.1 Eoscon Parameters

The usage of **eoscon** parameters for this model is:

eoscon(1) = ρ (material mass density).

eoscon(2)- eoscon(25) are unused constants.

17.1.2 Elastic Constants for 3-D Applications

This constitutive model for fully anisotropic elasticity has 21 independent elastic moduli which form the basis for the anisotropic orthotropic constitutive tensor shown below:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{31} \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \cdot \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \end{bmatrix}$$

While there are 36 coefficients listed in the above equation, only 21 are independent due to the major symmetry of the elastic tensor $C_{ij} = C_{ji}$. By appropriate specification of the moduli, one can achieve as special cases orthotropic, transversely isotropic, and isotropic elasticity.

For three-dimensional applications, the 21 constants are input to FENDAC in **five lines of 5f10 format:**

$C_{11}, C_{22}, C_{33}, C_{44}, C_{55}$

$C_{66}, C_{56}, C_{46}, C_{45}, C_{36}$

$C_{35}, C_{34}, C_{26}, C_{25}, C_{24}$

$C_{23}, C_{16}, C_{15}, C_{14}, C_{13}$

C_{12}

17.1.3 3-D Material Directors

The elastic constitutive tensor above is typically input with some assumption of material orientation. To allow the material to have an orientation other than perfectly aligned with a global Cartesian Coordinate system, material directors $\boldsymbol{\ell}_1$,$\boldsymbol{\ell}_2$, $\boldsymbol{\ell}_3$ embedded in the global coordinate system $\mathbf{e}_1$, $\mathbf{e}_2$, $\mathbf{e}_3$ are introduced. This allows FENDAC to model with anisotropic materials having arbitrary orientation. It should be noted, however, that each each material group has only a single orientation as specified by the directors.

The input of the director components is performed **on a single line of format 9f10**.

$\ell_{11}, \ell_{12}, \ell_{13}, \ell_{21}, \ell_{22}, \ell_{23}, \ell_{31}, \ell_{32}, \ell_{33}$

17.1.4 Elastic Constants for 2-D Applications

This constitutive model for fully anisotropic elasticity has 10 independent elastic moduli which form the basis for the anisotropic orthotropic constitutive tensor shown below:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} \\ C_{21} & C_{22} & C_{23} & C_{24} \\ C_{31} & C_{32} & C_{33} & C_{34} \\ C_{41} & C_{42} & C_{43} & C_{44} \end{bmatrix} \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{12} \end{bmatrix}$$

While there are 16 coefficients listed in the above equation, only 10 are independent due to the major symmetry of the elastic tensor $C_{ij} = C_{ji}$. As for the 3-D case, by appropriate specification of the moduli, one can achieve as special cases orthotropic, tranversely isotropic, and isotropic elasticity.

For two-dimensional applications, the 10 constants are input to FENDAC in **two lines of 5f10 format:**

$C_{11}, C_{22}, C_{33}, C_{44}, C_{34}$

$C_{24}, C_{23}, C_{14}, C_{13}, C_{12}$

17.1.5 2-D Material Directors

The material directors $\boldsymbol{\ell}_1$,$\boldsymbol{\ell}_2$ are embedded in the 2-D global coordinate system $\mathbf{e}_1$, $\mathbf{e}_2$. This allows FENDAC to model with anisotropic materials having arbitrary orientation in two-dimensions. As for the 3-D case, it should be noted that each each material group has only a single orientation as specified by the directors.

The input of the director components is performed **on a single line of format 4f10.**

$\ell_{11}, \ell_{12}, \ell_{21}, \ell_{22}$

17.2 MODEL #2: LINEAR ISOTROPIC/KINEMATIC HARDENING J_2 ELASTOPLASTICITY.

17.2.1 Eoscon Parameters

The usage of **eoscon** parameters for this model are:

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	Y_0	Initial yield stress
eoscon(3)	β	isotropic/kinematic hardening fraction
		$0 \leq \beta \leq 1$
eoscon(4)	H	plastic hardening shear modulus
eoscon(5)	FTOL	yield criteria tolerance
eoscon(6)	τ	time constant for Duvaut-Lions viscoplasticity
		if($\tau \leq 0$) inviscid response
		elseif($\tau > 0$) viscous response
eoscon(7)–eoscon(25)	—	Unused

The usage of material parameters is described by presentation of the model's rate form for a small-strain, small-rotation implementation:

$$\dot{\boldsymbol{\sigma}} = \mathbf{D} : (\dot{\boldsymbol{\epsilon}} - \dot{\boldsymbol{\epsilon}}^p) \tag{17.2.1}$$

$$\phi(\boldsymbol{\sigma}, \mathbf{q}) = \sqrt{\boldsymbol{\eta}' : \boldsymbol{\eta}'} - \kappa(\overline{e}^p), \tag{17.2.2}$$

in which

$$\kappa(\overline{e}^p) \equiv \sqrt{\frac{2}{3}}(Y_o + \beta \overline{e}^p) \tag{17.2.3}$$

$$\dot{\overline{e}}^p = \sqrt{\frac{2}{3}}\dot{\gamma} \tag{17.2.4}$$

$$\boldsymbol{\eta} \equiv \boldsymbol{\sigma} - \mathbf{q} \tag{17.2.5}$$

$$\boldsymbol{\eta}' = \mathbf{I}_{dev} : \boldsymbol{\eta} \tag{17.2.6}$$

$$\dot{\boldsymbol{\epsilon}}^p = \dot{\gamma}\frac{\partial \phi}{\partial \boldsymbol{\sigma}} = \dot{\gamma}\mathbf{n} \tag{17.2.7}$$

$$\mathbf{n} \equiv \frac{\boldsymbol{\eta}'}{||\boldsymbol{\eta}'||} \tag{17.2.8}$$

$$\dot{\mathbf{q}} = \frac{2}{3}(1 - \beta) H \dot{\boldsymbol{\epsilon}}^p. \tag{17.2.9}$$

Above, **D** represents a constant isotropic elastic tensor and ϕ represents the translating yield surface in stress space, where **q** is the center of the elastic domain in deviatoric space. In addition, the

Kuhn-Tucker and plastic consistency conditions which distinguish between loading and unloading states are written:

$$\phi \leq 0\,; \quad \dot{\gamma} \geq 0\,; \quad \dot{\gamma}\phi = 0, \quad \dot{\gamma}\dot{\phi} = 0. \tag{17.2.10}$$

The model is implemented using a fully implicit backward Euler integration algorithm along with consistent tangent operators.

17.2.2 Elastic Constants for 2-D and 3-D Applications

Input of the elastic constitutive tensor for this model follows the general format prescribed in Section 17.1.2 for 3-D applications, and in Section 17.1.4 for 2-D applications. The prescribed elastic constitutive tensor should be isotropic.

17.2.3 Material Directors

Due to the initial isotropy of this elasto-plasticity model, no material directors are required.

17.3 MODEL #3: DRUCKER-PRAGER ELASTOPLASTICITY WITH TENSION CAP

17.3.1 Eoscon Parameters

This Drucker-Prager model features:

(a) an associated flow rule,

(b) linear kinematic hardening with deviatoric plastic strain

(c) linear isotropic hardening with volumetric plastic strain

(d) a planar tension cutoff.

Eoscon parameters for the model are as listed below:

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	Y_0	Yield stress at $I_1 = 0; \overline{e}^p = 0$
eoscon(3)	B	$B = \frac{\partial\kappa}{\partial I_1}$ $(B \geq 0)$
eoscon(4)	H	plastic shear modulus (H $\geq$ 0)
eoscon(5)	FTOL	Relative tolerance for yield criterion.
eoscon(6)	A	Plastic bulk modulus (A $\geq$ 0)
eoscon(7)–eoscon(25)	—	Unused

The usage of material parameters is described by presentation of the model's rate form for a small-strain, small-rotation implementation:

$$\dot{\boldsymbol{\sigma}} = \mathbf{D} : (\dot{\boldsymbol{\epsilon}} - \dot{\boldsymbol{\epsilon}}^p) \tag{17.3.1}$$

$$\phi(\boldsymbol{\sigma}, \mathbf{q}) = \sqrt{\boldsymbol{\eta}' : \boldsymbol{\eta}'} - \kappa(Y_0, \overline{e}^p, I_1), \tag{17.3.2}$$

in which

$$\kappa(Y_o, \overline{e}^p, I_1) \equiv \sqrt{\frac{2}{3}} Y_o + B \cdot I_1 + A \cdot \overline{e}^p \tag{17.3.3}$$

$$\dot{\boldsymbol{\epsilon}}^p = \dot{\gamma} \frac{\partial \phi}{\partial \boldsymbol{\sigma}} \tag{17.3.4}$$

$$\dot{\overline{e}}^p = \mathrm{tr}\{\boldsymbol{\varepsilon}^{\mathrm{p}}\} \tag{17.3.5}$$

$$\boldsymbol{\eta} \equiv \boldsymbol{\sigma} - \mathbf{q} \tag{17.3.6}$$

$$\boldsymbol{\eta}' = \mathbf{I}_{dev} : \boldsymbol{\eta} \tag{17.3.7}$$

$$\dot{\mathbf{q}} = \frac{2}{3} \dot{\boldsymbol{\epsilon}}^p. \tag{17.3.8}$$

Above, $\mathbf{D}$ represents a constant isotropic elastic tensor and ϕ represents the translating yield surface in stress space, where $\mathbf{q}$ is the center of the elastic domain in deviatoric space. In addition, the

Kuhn-Tucker and plastic consistency conditions which distinguish between loading and unloading states are written:

$$\phi \leq 0\,; \quad \dot{\gamma} \geq 0\,; \quad \dot{\gamma}\phi = 0, \quad \dot{\gamma}\dot{\phi} = 0. \tag{17.3.9}$$

The model is implemented using a fully implicit backward Euler integration algorithm along with consistent tangent operators.

17.3.2 Elastic Constants for 2-D and 3-D Applications

Input of the elastic constitutive tensor for this model follows the general format prescribed in Section 17.1.2 for 3-D applications, and in Section 17.1.4 for 2-D applications. The prescribed elastic constitutive tensor should be isotropic.

17.3.3 Material Directors

Due to the initial isotropy of this elasto-plasticity model, no material directors are required.

17.4 MODEL #7: NON-SMOOTH THREE SURFACE TWO INVARIANT CAP MODEL

17.4.1 Eoscon Parameters

This elasto-plasticity model features:

(a) a linear-exponential failure envelope,
(b) a hardening elliptical compression cap surface,
(c) a planar tension cutoff surface, and
(d) linear deviatoric kinematic hardening.

Eoscon parameters for the model are as listed below.

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	α	Yield stress at $I_1 = 0$;
eoscon(3)	β	Exponential factor in failure envelope
eoscon(4)	θ	Linear slope coefficient for failure envelope
eoscon(5)	R	Shape factor for elliptic compression cap
eoscon(6)	λ	Subtracted factor in failure envelope.
eoscon(7)	W	Maximum compressive plastic strain
eoscon(8)	D	Exponential factor (cap hardening modulus)
eoscon(9)	κ_o	Initial cap parameter value
eoscon(10)	T	I_1 tension cutoff value
eoscon(11)	H	Twice the plastic shear modulus (kinematic hardening)
eoscon(12)	τ	Relaxation time for viscoplastic behavior
eoscon(13)	FTOL	Yield criterion tolerance (relative)

The usage of these material parameters is described by presentation of the model's rate form for a small-strain, small-rotation implementation. A more detailed description of the model's features can be found in Simo et al (1988). This model is appropriate for pressure-sensitive, isotropic porous materials such as soils, concretes, and grouts, as examples. An especially attractive feature of the pressure sensitive model is that it permits both dilatant and compactive plastic strains; loading on the failure envelope and tension cutoff surfaces leads to accumulation of dilatant plastic strains, while loading on the compression cap surface gives rise to compactive plastic strains (Figure 17.4.1). Since the model employed here is a slight variation of that presented Simo et al (linear kinematic hardening has been added), its full rate constitutive relations are written below:

The stress rate for the model is given by

$$\dot{\boldsymbol{\sigma}} = \mathbf{D} : (\dot{\epsilon} - \dot{\epsilon}^p) \tag{17.4.1}$$

subject to three yield constraints:

$$\phi_1(\boldsymbol{\sigma}, \mathbf{q}) = \|\mathbf{s} - \boldsymbol{\xi}\| - F_e(I_1) \leq 0 \tag{17.4.2.a}$$

$$\phi_2(\boldsymbol{\sigma}, \mathbf{q}, \kappa) = \|\mathbf{s} - \boldsymbol{\xi}\| - F_c(I_1, \kappa) \leq 0 \tag{17.4.2.b}$$

$$\phi_3(\boldsymbol{\sigma}) = -T + I_1 \leq 0, \tag{17.4.2.c}$$

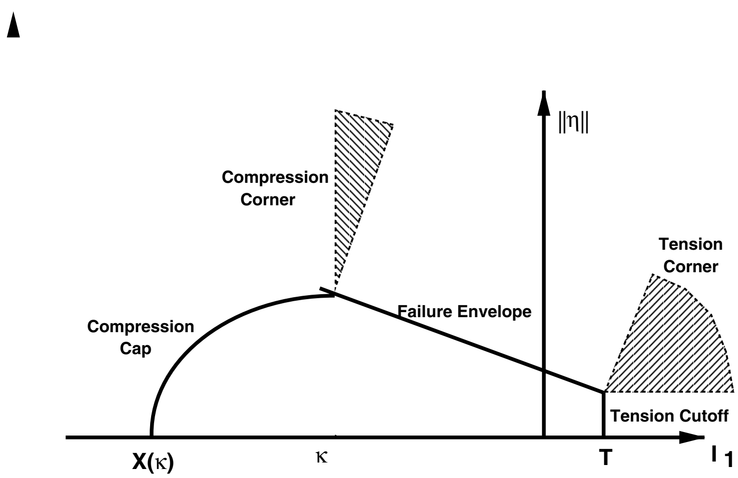


Figure 17.4.1: Non-smooth three-surface two-invariant elasto-plasticity model.

in which, the following definitions hold

$$
\begin{aligned}
\mathbf{s} &= \mathbf{I}_{dev} : \boldsymbol{\sigma} \\
\boldsymbol{\xi} &= \mathbf{I}_{dev} : \mathbf{q} \\
\boldsymbol{\eta} &= \mathbf{s} - \boldsymbol{\xi} \\
I_1 &= \mathrm{tr}\{\boldsymbol{\sigma}\}.
\end{aligned}
$$

The linear-exponential failure envelope surface F_e and the elliptical compression cap surface F_c have the following respective forms:

$$F_e(I_1) = \alpha - \theta I_1 - \lambda \exp(\beta I_1) \qquad \kappa \leq I_1 \leq T \tag{17.4.3.a}$$

$$F_c(I_1, \kappa) = \left[F_e^2(\kappa) - \frac{(I_1 - \kappa)^2}{R^2} \right]^{\frac{1}{2}} \qquad X(\kappa) \leq I_1 \leq \kappa \tag{17.4.3.b}$$

in which $X(\kappa) = \kappa - RF_e(\kappa)$ and in which κ is the so called *cap parameter*.

The flow rule for this model is associated and is expressed by Koiter's generalized form

$$\dot{\boldsymbol{\epsilon}}^p = \sum_{\alpha=1}^{3} \dot{\gamma}^\alpha \frac{\partial \phi_\alpha}{\partial \boldsymbol{\sigma}} \tag{17.4.4}$$

while the hardening laws for the model can be written as follows:

$$\dot{\kappa} = \begin{cases} 0 & \text{if } tr\dot{\boldsymbol{\epsilon}}^p \geq 0,\ I_1 = \kappa,\ \dot{\phi}_1 = \dot{\phi}_2 = 0 \\ \min \left[h'(\kappa) tr(\dot{\boldsymbol{\epsilon}}^p); \frac{||\dot{\boldsymbol{\eta}}||}{F_e'(\kappa)} \right] & \text{if } I_1 = \kappa,\ \dot{\phi}_1 = 0,\ \dot{\phi}_2 < 0 \\ h'(\kappa) tr(\dot{\boldsymbol{\epsilon}}^p) & \text{otherwise} \end{cases} \tag{17.4.5}$$

$$\dot{\mathbf{q}} = H \mathbf{I}_{dev} : \dot{\boldsymbol{\epsilon}}^p. \tag{17.4.6}$$

The variable tangent hardening modulus $h'(\kappa)$ for the cap parameter takes the exponential form

$$h'(\kappa) = \frac{exp\,[-DX(\kappa)]}{WD\,[1 + RF_e'(\kappa)]}. \tag{17.4.7}$$

The Kuhn-Tucker loading/unloading criteria and the plastic consistency conditions are written respectively

$$\dot{\gamma}^\alpha \geq 0 \quad ; \quad \phi_\alpha \leq 0 \quad ; \quad \dot{\gamma}^\alpha \phi_\alpha = 0 \qquad \text{for} \quad \alpha = 1, 2, 3 \tag{17.4.8}$$

$$\dot{\gamma}^\alpha \dot{\phi}_\alpha = 0 \qquad \text{for} \quad \alpha = 1, 2, 3. \tag{17.4.9}$$

The integration algorithm for this model is the fully implicit Backward Euler method implemented by Simo et al (1988) along with the recommendations made by Borja and Lee (1990) for the cap surface. Consistent tangent operators are used wherever possible, but special approximate tangent operators are required in the corner regions where two surfaces are active simultaneously, and a *symmetrized* consistent tangent operator is utilized when the compression cap alone is active.

17.4.2 Elastic Constants for 2-D and 3-D Applications

Input of the elastic constitutive tensor for this model follows the general format prescribed in Section 17.1.2 for 3-D applications, and in Section 17.1.4 for 2-D applications. When using this model, the prescribed elastic constitutive tensor should be isotropic.

17.4.3 Material Directors

Due to the initial isotropy of this elasto-plasticity model, no material directors are required.

17.5 MODEL #8: SMOOTH 3-SURFACE, 2-INVARIANT CAP MODEL

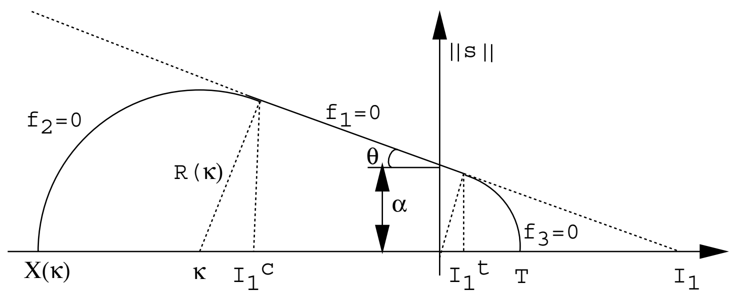


Figure 17.5.1: Smooth three-surface two-invariant elasto-plastic cap model.

17.5.1 Eoscon Parameters

This elasto-plasticity model features:

(a) a linear Drucker-Prager failure envelope,
(b) a hardening circular compression cap surface,
(c) a non-hardening circular tension cap surface, and
(d) linear deviatoric kinematic hardening.

Eoscon parameters for the model are as listed below.

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	α	Yield stress at $I_1 = 0$;
eoscon(3)	θ	Linear slope coefficient for failure envelope
eoscon(4)	W	Maximum compressive plastic strain
eoscon(5)	D	Exponential factor (cap hardening modulus)
eoscon(6)	κ_o	Initial cap parameter value
eoscon(7)	H	Twice the plastic shear modulus (kinematic hardening)
eoscon(8)	τ	Relaxation time for viscoplastic behavior
eoscon(9)	FTOL	Yield criterion tolerance (relative)

The usage of these material parameters is described by presentation of the model's rate form for a small-strain, small-rotation implementation. A more detailed description of the model's features can be found in Seo and Swan (1997). This model is appropriate for pressure-sensitive, isotropic porous materials such as soils, concretes, and grouts, as examples. An especially attractive feature of the pressure sensitive model is that it permits both dilatant and compactive plastic strains; loading on the failure envelope and tension cutoff surfaces leads to accumulation of dilatant plastic strains, while loading on the compression cap surface gives rise to compactive plastic strains (Figure 17.5.1). The integration algorithm for this model is the fully implicit Backward Euler method implemented by Seo and Swan (1997). Consistent tangent operators are used to facilitate rapid convergence in global force-balance iterations.

17.5.2 Elastic Constants for 2-D and 3-D Applications

Input of the elastic constitutive tensor for this model follows the general format prescribed in Section 17.1.2 for 3-D applications, and in Section 17.1.4 for 2-D applications. When using this model, the prescribed elastic constitutive tensor should be isotropic.

17.5.3 Material Directors

Due to the initial isotropy of this elasto-plasticity model, no material directors are required.

17.6 MODEL #10: HILL'S ORTHOTROPIC ELASTOPLASTICITY WITH TENSORAL HARDENING

This distortional plasticity model is appropriate for modeling anisotropic materials such as nonfrictional composites. A complete description of this model's capabilities and limitations is provided in Swan and Cakmak (1994). The yield surface for this model is a generalized ellipsoid in six-dimensional stress space which is permitted to translate via linear kinematic hardening. Within a given element group, FENDAC will expect to read in NUMAT material property data sets. For each material group, the properties data is entered in five different modules:

a. Material property set identifier number;

b. Eoscon property data which contains the radii of the yield surface in deviatoric stress space;

c. Orthotropic elasticity tensor;

d. Orthotropic plastic hardening tensor;

e. Material directors.

The input format for these modules is described below.

17.6.1 Material Property Identifier

The material property set identifier is an integer that should be entered on a separate line in **I5** format.

17.6.2 Eoscon Parameters

The usage of **eoscon** parameters for this model is different for 3D and 2D applications.

For 3D applications, the model can be used in either elastoplastic mode or in visco-elastic-plastic mode. The eoscon parameters for the model are entered on a single line in **25F10** format and are as listed below.

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	R_{11}	S_{11} yield radius
eoscon(3)	R_{22}	S_{22} yield radius
eoscon(4)	R_{33}	S_{33} yield radius
eoscon(5)	R_{23}	S_{23} yield radius
eoscon(6)	R_{31}	S_{31} yield radius
eoscon(7)	R_{12}	S_{12} yield radius
eoscon(9)	FTOL	Yield criterion tolerance (relative)
eoscon(10)	ROTATE	Elasticity Tensor Rotation Indicator
		If > 0, ...
eoscon(11)	τ	Relaxation time for viscoplastic behavior

For 2D applications, this model can only be used in elastoplastic mode. That is, it cannot presently accommodate viscoplastic behavior. Eoscon parameters for the model are as listed below.

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	R_{11}	S_{11} yield radius
eoscon(3)	R_{22}	S_{22} yield radius
eoscon(4)	R_{33}	S_{33} yield radius
eoscon(5)	R_{12}	S_{12} yield radius
eoscon(6)	FTOL	Yield criterion tolerance (relative)

17.6.3 Elasticity Tensor

For 3D applications, the input of the 21 components of the orthotropic elasticity tensor is as outlined in Section 17.1.2, whereas for 2D applications the 10 components of the elasticity tensor are entered as described in Section 17.1.4.

17.6.4 Plastic Hardening Tensor

The input of the linear kinematic hardening tensor is completely analogous to that of the elasticity tensor. For 3D applications, the input format of the 21 components of the hardening tensor is as outlined in Section 17.1.2, whereas for 2D applications the 10 components of the hardening tensor are entered as described in Section 17.1.4.

17.6.5 Material Directors

The elastic constitutive tensor above is typically input with some assumption of material orientation. To allow the material to have an orientation other than perfectly aligned with a global Cartesian Coordinate system, material directors $\boldsymbol{\ell}_1$,$\boldsymbol{\ell}_2$, $\boldsymbol{\ell}_3$ embedded in the global coordinate system $\mathbf{e}_1$, $\mathbf{e}_2$, $\mathbf{e}_3$ are introduced. This allows FENDAC to model with anisotropic materials having arbitrary orientation. It should be noted, however, that each each material group has only a single orientation as specified by the directors.

In three-dimensions the input of the director components is performed **on a single line of format 9f10**.

$\ell_{11}, \ell_{12}, \ell_{13}, \ell_{21}, \ell_{22}, \ell_{23}, \ell_{31}, \ell_{32}, \ell_{33}$

In two dimensions, the material directors $\boldsymbol{\ell}_1$,$\boldsymbol{\ell}_2$ are embedded in the 2-D global coordinate system $\mathbf{e}_1$, $\mathbf{e}_2$. This allows FENDAC to model with anisotropic materials having arbitrary orientation in two-dimensions. As for the 3-D case, it should be noted that each each material group has only a single orientation as specified by the directors.

The input of the director components is performed **on a single line of format 4f10.**

$\ell_{11}, \ell_{12}, \ell_{21}, \ell_{22}$

17.7 MODEL #11: STANDARD LINEAR SOLID VISCOELASTICITY MODEL

This standard solid isotropic viscoelasticity model can be considered as comprised of a volumetric or bulk part, and a deviatoric part. The form of the constitutive equation is

$$[\mathbf{E}_1 + \mathbf{E}_2] : \dot{\epsilon} + \mathbf{E}_1 : \boldsymbol{\eta}^{-1} : \mathbf{E}_2 \epsilon = \dot{\boldsymbol{\sigma}} + \mathbf{E}_1 : \boldsymbol{\eta}^{-1} : \boldsymbol{\sigma}$$

where $\mathbf{E}_1$ and $\mathbf{E}_2$ are linear isotropic elasticity tensors and $\boldsymbol{\eta}$ is a linear isotropic viscosity tensor:

$$\mathbf{E}_1 = k_1 \mathbf{1} \otimes \mathbf{1} + \mu_1 \mathbf{I}_{dev}$$
$$\mathbf{E}_2 = k_2 \mathbf{1} \otimes \mathbf{1} + \mu_2 \mathbf{I}_{dev}$$
$$\boldsymbol{\eta} = \eta_b \mathbf{1} \otimes \mathbf{1} + \eta_s \mathbf{I}_{dev}.$$

Input for this model is broken into three separate modules:

a. Material property set identifier number;

b. Eoscon property data;

c. The isotropic elasticity tensor;

The input format for these modules is described below.

17.7.1 *Material Property Identifier*

The material property set identifier is an integer that should be entered on a separate line in **I5** format.

17.7.2 *Eoscon Parameters*

The usage of **eoscon** parameters for this model is identical for 3D and 2D applications. In addition to a mass density, the model has six parameters: k_1, μ_1, k_2, μ_2, η_b and η_s. The parameters k_1, μ_1 are entered with the elasticity tensor, and the parameters k_2, μ_2, η_b and η_s are entered in the eoscon vector.

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	k_1	viscoelastic bulk stiffness
eoscon(3)	μ_1	viscoelastic shear stiffness
eoscon(4)	k_2	elastic bulk stiffness
eoscon(5)	μ_2	elastic shear stiffness
eoscon(6)	η_b	bulk viscosity
eoscon(7)	η_s	shear viscosity

17.7.3 *Elasticity Tensor*

For this model, there is no need to input the elasticity tensor since it is generated from the constants k_2 and μ_2 entered via the estcon array.

17.7.4 *Material Directors*

Since this model is isotropic, no material directors are required.

17.8 MODEL #12: ISOTROPIC, HYPERELASTIC SOLID

This hyperelastic constitutive model has decoupled volumetric energy U and deviatoric strain energy $\bar{W}$ functions of the form:

$$U(J) = \frac{1}{2}K\left[\frac{1}{2}(J^2 - 1) - \ln(J)\right]$$
$$\bar{W} = \frac{1}{2}\mu\,\mathbf{tr}[\bar{\mathbf{b}} - 3]$$

where:

J	is the determinant of **F**, the deformation gradient;
K	is a material constant which functions as a bulk modulus;
μ	is a material constant which functions as a shear modulus;
$\bar{\mathbf{b}}$	is $J^{\frac{-2}{3}}\mathbf{b}$, where $\mathbf{b} = \mathbf{F}\mathbf{F}^{\mathbf{T}}$ is the left Cauch-Green Deformation tensor;

For this model, the Kirchoff stress $\boldsymbol{\tau}$ is therefore given as:

$$\boldsymbol{\tau} = JU'(J)\mathbf{1} + 2\text{dev}\frac{\partial\bar{W}}{\partial\bar{\mathbf{b}}}.$$

For each material property group within an element group there must be a definition of material properties. For each material property group there will be:

a. Material property set identifier number; and

b. Eoscon property data.

The input format for these modules is described below.

17.8.1 Material Property Identifier

The material property set identifier is an integer that should be entered on a separate line in **I5** format.

17.8.2 Eoscon Parameters

There are only three **eoscon** parameters to be entered for this model.

Parameter	Constant	Description
eoscon(1)	ρ	Material mass density
eoscon(2)	K	Bulk modulus
eoscon(3)	μ	Shear modulus

18. BIBLIOGRAPHY

[1] C.C. Swan, "Techniques for stress and strain controlled homogenization of inelastic periodic composites," Computer Methods in Applied Mechanics and Engineering **117** 249–267 (1994).

[2] C.C. Swan, "Iterative and direct techniques for 3-d homogenization of periodic elastoplastic composites using super vector-parallel computers," in Advances in Non-linear Finite Element Methods, B.H.V. Topping and M. Papadrakakis (Eds.) Civil-Comp. Press, 279–289 (1994).

[3] J.C. Simo, J.W. Ju, K.S. Pister, and R.L. Taylor, "Assessment of cap model: consistent return algorithms and rate-dependent extension," Journal of Engineering Mechanics bf 114 (2) 191–218 (1988).

[4] R.I. Borja and S.R. Lee, "Cam-clay plasticity, part I: Implicit integration of elasto-plastic constitutive relations," Computer Methods in Applied Mechanics and Engineering **78** 49–72 (1990).

[5] Y.-K. Seo, Computational methods for elastoplastic slope stability analysis with seepage, *(Ch. 5: A smooth cap plasticity model for soils: integration algorithm, parameter estimation and applications*, Ph.D dissertation, University of Iowa (1998).

[6] C.C. Swan and A.S. Cakmak, "A hardening orthotropic plasticity model for non-frictional composites: rate formulation and integration algorithm," International Journal for Numberical Methods in Engineering **37** 839–860 (1994).

[7] C.C. Swan and J.S. Arora, "Topological optimization of material layout in structured composites of high stiffness and high strength," Structural Optimization **13-1** 45-57 (1997).

[8] C.C. Swan and I. Kosaka, "Voigt-Reuss topology optimization for structures with linear elastic material behaviours," International Journal for Numerical Methods in Engineering **40** 3033-3057 (1997).

[9] C.C. Swan and I. Kosaka, "Voigt-Reuss topology optimization for structures with nonlinear material behaviours," International Journal for Numerical Methods in Engineering **40** 3785-3814 (1997).

[10] A.R. Mijar, C.C. Swan, J.S. Arora, and I. Kosaka, "Continuum topology optimization for concept design of frame bracing systems," Journal of Structural Engineering **124-5** 541-550 (1998).

[11] I. Kosaka and C.C. Swan, "A symmetry reduction method for continuum structural topology optimization," Computers & Structures **70-1** 47–61 (1999).

[12] C.C. Swan and Y.-K. Seo, "Limit state analysis of earthen slopes using dual continuum/fem approaches," International J. for Numerical and Analytical Methods in Geomechanics **23** 1359-1371 (1999).

[13] Y.-K. Seo and C.C. Swan, "Stability analysis of embankments on saturated soils using elasto-plastic porous medium models," *Journal of Geotechnical and Geoenvironmental Engineering* **127** No. 5 (2001).