REPORT NO.
UCB/SEMM-92/30

STRUCTURAL ENGINEERING
MECHANICS AND MATERIALS

DRAIN-3DX:
Base Program User Guide

BY

V. PRAKASH
G. H. POWELL
and
F. C. FILIPPOU

DECEMBER 1992

DEPARTMENT OF CIVIL ENGINEERING
UNIVERSITY OF CALIFORNIA
BERKELEY, CALIFORNIA

DRAIN - 3DX

Version 1.03

BASE PROGRAM USER GUIDE

December 1992

by

V. Prakash
G. H. Powell
and
F. C. Filippou

DRAIN-3DX
USER GUIDE SECTIONS

1. SUMMARY OF MODELING AND ANALYSIS PROCEDURES

1.1 STRUCTURE MODEL

Since the current version of the program uses an "in-core" equation solver and does not renumber the nodes for minimum storage, it is suitable only for relatively small structures.

The structure is modeled as a 3D assemblage of nonlinear elements connected at nodes.

Nodes are identified by number, and need not be numbered sequentially. Unless specified otherwise, each node has six degrees of freedom (global X, Y and Z translations and X, Y and Z rotations). If a node is restrained or slaved to another node it will have fewer than six DOFs.

The elements must be divided into groups. All elements in a group must be of the same type. However, all elements of a given type need not be in a single group. Elements must be numbered sequentially within each group. An element is identified by its group number and element number.

The structure mass is lumped at the nodes, and the mass matrix is diagonal. When a node is slaved to a master node, its masses are transformed to equivalent masses at the master node, and hence the master node may have a non-diagonal mass matrix. If so, the off-diagonal terms are ignored.

A viscous damping matrix that is proportional to the element stiffnesses and nodal masses can be specified (i.e., the form of this matrix is $C = \alpha M + \beta K$). Mass dependent damping introduces translational and/or rotational dampers at each node, with damping coefficients αM. Different values of α can be specified for each node if desired. Stiffness dependent damping introduces dampers in parallel with the elements. Different values of β can be specified for each element group. The damper stiffness, βK, for any element is constant and is based on the initial stiffness of the element. If an element has zero initial stiffness which subsequently increases (e.g., a support element with an initially open gap) it will always have a zero C matrix. If an element has a very large initial stiffness which subsequently decreases (e.g., a stiff support element which yields) it will always have a large C matrix. This is the same as "βK_0" damping in the original DRAIN-2D program. The "βK_T" option, where C is based on the current tangent stiffness, is not available in the new program.

The program documentation allows for both conventional nodes and "compound" nodes. A compound node consists of a main node plus a number of subnodes. Compound nodes have been included to make it simpler for complex connection elements to be defined in the future. Compound nodes have not yet been fully implemented, and are not currently used by any element types. Hence, they should not be defined.

1.2 ANALYSIS TYPES

Analyses of the following types can be performed.

Gravity : Static analysis for combined element loads and nodal loads. In the current version of the program the structure must remain linear for this type of analysis.

Static : Nonlinear static analysis; for nodal loads only.

Restore to Static State : At the end of a dynamic analysis, the structure will generally still be moving, and hence will not be in static equilibrium. This analysis restores the structure to a static equilibrium state.

Mode Shapes and Periods : Calculation of mode shapes and periods, in the initial state or any later state.

Response Spectrum : Linear analysis for specified X, Y and/or Z response spectra, using mode shapes and periods calculated for the initial state.

New Ground Acceleration : Nonlinear dynamic analysis for ground motions acceleration records for X, Y, Z translation and/or X, Y, Z rotation. All supports must move in-phase.

Resume Ground Acceleration : Continue the preceding ground acceleration analysis for an additional time segment. If desired, mode shapes and periods can be calculated between the two analyses.

New Ground Displacement : Nonlinear dynamic analysis for ground motions defined by displacement records for X, Y, Z translation and/or X, Y, Z rotation at support points, including different (out of phase) motions at different supports.

Resume Ground Displacement : Continue the preceding ground displacement analysis for an additional time segment. If desired, mode shapes and periods can be calculated between the two analyses.

New Dynamic Force : Nonlinear dynamic analysis for dynamic forces defined by force records for X, Y, Z translation and/or X, Y, Z rotation on the nodes.

Resume Dynamic Force : Continue the preceding dynamic force analysis for an additional time segment. If desired, mode shapes and periods can be calculated between the two analyses.

New Initial Velocity : Nonlinear dynamic analysis for specified initial nodal velocities. This analysis can be used to compute the response following an impact or to evaluate energy absorption capacity.

Resume Initial Velocity : Continue the preceding initial velocity analysis for an additional time segment. If desired, mode shapes and periods can be calculated between the two analyses.

A structure will typically be analyzed for several different loadings, carried out in a number of "analysis sessions". In any analysis session, analyses can be performed for any number of "analysis segments", each segment consisting of either a static load increment or the application of a dynamic load for a period of time. The analysis segments are numbered in sequence, with the initial state assigned a segment number of 0. The structure state at the end of any analysis segment can be saved on a permanent file, if desired.

In any analysis session, the beginning state for the first analysis segment can be any previously saved state (identified by its analysis segment number), not necessarily the last state. If there are several analysis segments in a session, the beginning state for each segment (except the first) is always the state at the end of the preceding segment. This feature allows unlimited flexibility in choosing load sequences. In particular, static and dynamic segments can be inter-mixed. However, a static gravity segment must begin from the initial state.

1.3 LOADS

Loads are input as *patterns* (for static loads) or as *records* (for dynamic loads). The loads for any analysis segment are defined as combinations of patterns or records. If desired, new patterns and records can be added in any analysis session.

Seven different load types can be specified, as follows :

Static Element Load Patterns : The element loads which make up any pattern are defined element group by element group. The types of load (gravity, prestress, etc.) which are allowed depend on the element type. Some element types do not have provisions for element loads. Element loads can be applied only in gravity analysis segments, and the behavior must be linear.

Static Nodal Load Patterns : Each pattern consists of loads applied on nodes. Static nodal loads can be applied in gravity and/or static analysis segments. The loading for any analysis segment can be any combination of the available patterns.

Ground Acceleration Records : Translational or rotational records can be specified. As many records as desired may be defined, but a maximum of six can be used for a ground acceleration analysis.

Ground Displacement Records : Ground displacement records can be specified for translations and/or rotations. It is important to note that there are two types of support, namely rigidly restrained and spring supported. Ground accelerations are applied at supports of both types, but ground displacements are applied only at spring supports. Out-of-phase effects can be produced either by specifying different records at different support points or by specifying different time delays. In this second procedure, each

displacement record defines a wave train which moves past the structure. As many records as desired may be defined but a maximum of thirty-six can be used in any analysis.

Dynamic Force Records : Dynamic force records consist of time-force pairs acting on nodes, and can be applied in dynamic force analyses. As many records as desired may be defined but a maximum of thirty-six can be used in any analysis.

Initial Velocity Patterns : Initial velocity patterns consist of translational or rotational nodal velocities. The initial velocities for analysis can be any combination of the available patterns, and the combination can be scaled to provide a given initial kinetic energy.

Response Spectra : Displacement, velocity or acceleration response spectra can be specified. As many response spectra as desired may be defined, but a maximum of three (in the X, Y and Z directions) can be used for a response spectrum analysis.

1.4 ANALYSIS PROCEDURE

In a static analysis the load will typically be applied in a number of steps. Within any step the program further selects a load *substep* size, by determining when the next stiffness change (event) occurs, and ending the substep at that event. The structure stiffness is then modified, and an analysis is performed for the next substep. The analysis segment ends when all of the load has been applied, or alternatively when a specified displacement is reached.

In a dynamic analysis the time step may be specified to be constant or variable. If the time step can vary, upper and lower error tolerances to control the step size must be specified. The program computes an error measure in each step. If this measure exceeds the upper tolerance in any step, the time step is reduced and the step is repeated. If the measure is less than the lower tolerance for some specified number of steps (typically 2), the time step is increased in the following step. The time step decrease and increase factors can be specified. They will typically be 0.5 and 2.0, respectively.

Other options are available for dynamic analysis, including (a) event computations within time steps (in which case each time step may be divided into substeps) and (b) corrections at the end of each time step to improve energy balance and/or equilibrium. For details on the dynamic step-by-step strategy, refer to the theoretical documentation.

1.5 P-DELTA EFFECTS

P-delta effects can be considered if desired. This is done by adding a geometric stiffness matrix to the stiffness matrix for each element, and accounting for P-delta effects in the resisting force computation. The geometric stiffness is changed at each event in a static analysis. It can be kept constant for dynamic analysis, or alternatively allowed to change. For details on P-delta effects, refer to the theoretical documentation.

None of the currently available elements accounts for true large displacement effects. This could be done. However, see Section 1.8.

1.6 EVENT OVERSHOOT TOLERANCES

Event overshoot tolerances can be defined for each element. The event factor is then calculated for the nominal event plus the tolerance. This can reduce the execution time of the program, by allowing several elements to yield or unload at once, so that the structure stiffness matrix is modified fewer times. Large overshoot tolerances, however, will lead to large unbalanced loads (see Section 1.8).

The overshoot tolerances specified with the element input data are nominal values, which can be increased or decreased later if desired. This is done by means of overshoot tolerance scale factors, which can be changed without re-entering the input data. Different scale factors can be specified for each element group, and for static and dynamic loading.

1.7 ENERGY BALANCE

Energy calculations can be performed, for both static and dynamic analyses. The calculation accounts for external work on the nodes, static elastic-plastic work on the elements, kinetic energy, and viscous damping work. If there is a significant energy unbalance, the analysis results are likely to be inaccurate.

1.8 EQUILIBRIUM ERROR (UNBALANCED LOADS)

The event-to-event solution strategy for static analysis is used because it is the most reliable and the easiest to use. If the elements all have mutlilinear action-deformation relationships, if small overshoot tolerances are specified, and if the element geometric stiffnesses do not change rapidly, the solution will stay close to the equilibrium path and there will be small unbalanced load errors. However, if some elements have curvilinear behavior, if large overshoots are allowed, or if the element axial forces (and hence their geometric stiffnesses) change rapidly, there may be significant unbalances. The program calculates the unbalances at the end of each load step, and applies them as corrections in the next step, but does not iterate on the unbalance. If the results show significant unbalanced loads at the end of an analysis, either (a) repeat the analysis with a larger number of load steps (in which case the unbalanced load correction is made more often), or (b) add a dummy static analysis with zero load (which, in effect, iterates on the unbalance).

1.9 STRUCTURE SECTIONS

Plane sections cutting through the structure may be specified, and the resultant normal force, shear force and overturning moment can be calculated for each section. The procedure for specifying structure sections may seem to be awkward, but is necessary for generality, and in most cases the required input data is easy to provide.

1.10 GENERALIZED DISPLACEMENTS

Up to 8 displacements can be combined to form a single-valued "generalized" displacement, for which results are output. For example, (a) an interstory drift can be obtained by specifying a generalized displacement as the horizontal displacement of the upper floor minus that of the lower floor, and (b) a more accurate measure of interstory deformation can be obtained by specifying a generalized displacement which calculates the shear distortion of the story.

1.11 RESULTS OUTPUT

Results for selected nodal displacements, element responses, structure sections and generalized displacements can be written to an output text file if desired. This file can then be printed. Mode shapes, periods and the results of response spectra analyses are also written to the output file.

Results are written to a number of binary files, for processing by a post-processing program (DRAIN-POST). This program is being developed but is not yet complete.

2. PERMANENT FILES

A number of permanent files are created for each problem. All files have names of the form PROBNAME.EXT, where PROBNAME is the problem name and EXT is a three character extension. The extensions have the following meanings.

ACC: A binary file containing ground acceleration records. As new records are input they are appended to the file.

DIS: A binary file containing ground displacement records.

ECH: A text file containing an echo of the input data plus an analysis log. The analysis log consists of event and unbalanced load information. If the file exists at the beginning of any analysis session, the new output is appended.

EXX: A binary post-processing file containing envelope results for analysis segment XX. These files are used for post-processing of results by DRAIN-POST. Note that a separate file is created for each analysis segment.

FRC: A binary file containing dynamic force records.

ELD: A binary file containing element load patterns.

GEO: A binary file containing the coordinates of all nodes, the node numbers for all elements, and other data. This file is used by DRAIN-POST.

LST: A text file listing the analysis segments which have been completed, and showing the segments for which .EXX, .MXX, .RXX and .SXX files have been created.

MXX: A binary file containing mode shapes and periods for analysis segment XX.

OUT: A text file containing results for the last analysis session. The file contains time histories and envelopes of nodal displacements, element results, structure section forces and generalized displacements. If the file exists at the beginning of any analysis session, the new output is appended.

RXX: A binary post-processing file containing results for analysis segment XX. These files are used for post processing of results by DRAIN-POST. Note that a separate file is produced for each analysis segment. If the file exists at the beginning of any analysis session, the new output is appended.

SLO: A text file containing a solution log of the last session. This log consists of energy balance and unbalanced load information. If the file exists at the beginning of any analysis session, the new output is appended.

SPC: A binary file containing earthquake response spectra.

STA: A binary file containing static nodal load patterns.

SXX: A binary file containing the structure state at the end of analysis segment XX. These files are used for restarting.

UXX: A binary file containing unit spectral response results for analysis segment XX.

VEL: A binary file containing initial velocity patterns.

For detailed descriptions of the binary files, see the program design documentation.

3. NOTES ON RUNNING ANALYSES

3.1 SUGGESTED PROCEDURE FOR ORGANIZING FILES

1. Create a subdirectory of the EXAMPLES directory for each new problem. For example, if the problem name is SAMPLE, create a subdirectory SAMPLE or SAMPLE.DIR.
2. Set up input files for the problem in the problem directory. For example, the file SAMPLE-G.INP might be the geometry input data, SAMPLE-S.INP might be the input data for static analysis, and SAMPLE-D.INP might be the input .data for dynamic analysis.
3. To run any input file, copy it to DRAIN.INP. For example, in DOS give the command COPY SAMPLE-G.INP DRAIN.INP.
4. The executable program is in the directory ..\.\EXE. In DOS the program can be run by giving the command ..\.\EXE\D3DX. It is easier, however, to create a batch file DRAIN.BAT (which contains the single line ..\.\EXE\D3DX) and give the command DRAIN. The file DRAIN.BAT can be copied from the EXAMPLES directory. All files created by the program will be in the problem directory.

3.2 HINTS AND THINGS TO WATCH FOR

3.2.1 NOTABS

Some text editors replace successive blanks with tab characters. Be sure to save DRAIN.INP files without tabs.

3.2.2 EXISTING FILES

Certain files (e.g., the .ECH and .OUT files) must be new files at the beginning of a new problem (*START). If one of these files already exists for a new problem, the analysis stops. If the file is to be kept it must be renamed. If not it must be erased. If these files already exist for a continuing analysis (*RESTART) the new output is appended.

3.2.3 LONG OUTPUT FILES

If the results of an analysis session are output on a .OUT text file, this can be a very large file. To avoid large files, it is wise to run large dynamic analyses in a number of analysis sessions, and to rename the .OUT file after each session. This produces a number of shorter files.

If results are output to binary files (for processing by the forthcoming DRAIN-POST program), it may be wise to run large dynamic analyses in several segments (not several sessions), since a separate file (e.g. .RXX) is created for each segment.

3.2.4 NODE NUMBERING

Nodes do not have to be numbered sequentially. It is usually a good idea to set up grid lines, with nodes at grid line intersections, and to combine grid line numbers to get node numbers. For an example see the DRAIN-2DX HMRFRM example problem.

The input data allows up to 10 digits for node numbers. This allows 3 characters for each of three grid lines in a 3D (i.e., DRAIN-3DX).model.

The program does not currently renumber nodes, but orders them in increasing numerical order. For efficiency, it is desirable to number the nodes to minimize the band width. This will usually happen naturally if grid lines are used.

It is usually easier to generate nodes and elements if grid lines are used.

3.2.5 DATA CHECKING AND RESTART

The lengths of input files, and the execution time, can be reduced if the *RESTART option is used. A recommended procedure is as follows.

1. Use *START with the execution code (KEXE) set = 1, until the geometry and property input data is correct. Note that the program usually will continue executing if an input error is found, and may stop or crash at a later time. Set the input echo code to 1, so that you will know how far the program has progressed through the DRAIN.INP file, and check the .ECH file for error and warning messages. Load patterns and records may be input at this stage, or may be input later.
2. When the data is correct, set KEXE = 0 and run the program to save the geometry and property data (this will be the initial state, or State 0).
3. Prepare subsequent DRAIN.INP files with load and analysis data only, restarting from the previously saved state (State 0, or some later saved state).

For an example, see the HMRFRM example problem. Note that new loads can be added, and control parameters can be changed, in any session. However, the geometry and properties can not be changed.

3.2.6 POST-PROCESSING

The post-processor program is not yet complete. Hence, to plot results it is necessary to extract data tables from the .OUT file, using a text editor, and to plot the tables using, say, a spreadsheet program. See the DRAIN-2DX CNNFRM AND HMRFRM example problems for examples of tables which might be extracted from the .OUT file. This is, admittedly, a time consuming procedure.

3.2.7 CHECKING RESULTS

The .ECH file contains an input echo and the .ECH and .SLO files contain logs of the analysis steps. Be sure to check these files. In particular, check the unbalanced loads, in .ECH or .SLO. A large unbalance indicates an inaccurate analysis. It is wise to perform energy calculations (see the *START line), and to check .SLO to ensure that there is no serious energy error. A substantial energy error indicates an inaccurate analysis.

3.2.8 EVENT OVERSHOOT

The program uses an event-to-event strategy to solve the nonlinear problem, where an event corresponds to a significant change in stiffness. The program corrects for equilibrium errors, but it does not iterate on the unbalance (i.e., it does not perform Newton-type iteration).

It is usually wise to avoid large unbalances by turning on the event calculation option. To do this:

1. Set KEVE = 1 in *ELEMENTGROUP lines.
2. Specify reasonable overshoot tolerances as part of the element property data. If the overshoot tolerance is very small, an event will occur almost exactly when the element stiffness changes (e.g., at the nominal element strength or immedialtely after the element unloads), and equilibrium unbalances will be small. If the overshoot tolerance is large (or if KEVE = 0), there will be overshoot before the event occurs, and equilibrium unbalances can be large.
3. For static analysis, events are always calculated. For dynamic analysis, event calculation is an option, and the default is "no". To turn on event calculation for dynamic analysis, set KEVNT = 1 in the *PARAMETERS/DC line.

The overshoot defined with the element properties can be changed if desired, by use of the *PARAMETERS/F option. See the HMRFRM example problem, where a unit overshoot is specified with the element properties, and then analyses with two different values are performed. With a small overshoot the analysis is more accurate. With a larger overshoot there are fewer events (and hence the analysis takes a little less time), because two or more elements can change stiffness at the same event. However, the analysis is less accurate.

As already noted, if the elements all have mutlilinear action-deformation relationships, if small overshoot tolerances are specified, and if the element geometric stiffnesses do not change rapidly, the solution will stay close to the equilibrium path and there will be small equilibrium unbalances. However, if some elements have curvilinear behavior, if large overshoots are allowed, or if the element axial forces (and hence their geometric stiffnesses) change rapidly, there may be significant unbalances. The program calculates the unbalances at the end of each load step, and applies them as corrections in the next step, but does not iterate on the unbalance. If the results show significant unbalanced loads at the end of an analysis, either (a) repeat the analysis with a larger number of load steps (in which case the unbalanced load correction is made more often), or (b) add a dummy static analysis with zero load (which, in effect, iterates on the unbalance).

3.2.9 DYNAMIC ANALYSIS

Except for changing the amount of overshoot and/or changing the number of analysis steps, the analyst has little control over the static analysis strategy. However, there are more options for dynamic analysis, including the following.

1. Events may or may not be considered. It is usually recommended that events be considered (see the *PARAMETERS/DC data). Execution time may be saved by increasing the overshoot tolerances, using the *PARAMETERS/F option, but at the expense of accuracy.
2. Time steps may be constant or variable. The constant time step option can often be used. However, if there are gap-type elements, the variable step option should definitely be used. This is because gap closure involves impact, and a very small time step may be needed for a short period of time immediately after impact. The variable time step option is very effective for this purpose. It is recommended that the "midstep equilibrium error" ("static error") option be used, not the "impulse" option. See the *PARAMETERS/DA and /DT data.
3. The program includes options for "velocity correction" and "acceleration correction". These corrections may help in some cases, but it is generally recommended that they be omitted. See the *PARAMETERS/DC data.

3.2.10 COMMENTS IN INPUT DATA

The DRAIN.INP file can contain any number of comments (preceded by "!"). It is a good idea to use comments, since they are very helpful in explaining the input.

4. INPUT DATA FILE

The following sections describe the required input data. Except as noted, units must be consistent throughout.

This is a guide for the base program only. User guides for the elements are in separate reports.

The name of the input file must be DRAIN.INP (in upper case letters).

Data items are of type C, I or R. Type C is character data (FORTRAN "A" conversion), type I is integer, and type R is real (FORTRAN "F" or "E" conversion).

Comments can be added by using exclamation marks (!). If '!' is in the first column of an input line, the whole line is treated as a comment and is ignored. This is useful for long, multi-line comments. If '!' is not in the first column of the input line, the information to the right of '!' is a comment and is ignored. This is useful for shorter comments.

In the input file "separator lines" are used to divide the data into logical groups. The separators must be in upper case letters and must start in column 1.

Expanatory notes are at the end of this guide. "G" notes are referred to from several sections of the guide, and can be found in the GENERAL NOTES section. "S" notes are specific to individual sections of the guide, and are in separate sections identified by the section separators (e.g., *NODECOORDS : NOTES for notes on node coordinate specification).

The separators as follows.

***START/*RESTART** : Problem Initiation.

Specify problem name, initial state and parameters for the analysis session. *START is used for a new problem. *RESTART is used for an old problem.

***COMPOUND** : Compound Node Types

Define a compound node type. Compound node type information consists of the number of subnodes, their coordinates with respect to the main node and their degrees of freedom. Repeat for each compound node type.

***NODECOORDS** : Node Coordinates

Specify or generate coordinate locations of all nodes.

***NODETYPES** : Node Type Specification

Specify node type for each compound node.

***RESTRAINTS** : Restrained Nodes

***SLAVING** : Nodal Slaving Constraints

***MASSES** : Nodal Masses

***ELEMENTGROUP** : Element Group Definition

Define an element group. Repeat for each element group.

***SECTION** : Structure Section Definition

Define a structure section. Repeat for each structure section.

***GENDISP** : Generalized Displacement Definition

Define a generalized displacement. Repeat for each generalized displacement.

***RESULTS** : Results Output Specification

Specify the results that are to be written to the .RXX and .OUT files. These can be changed in each analysis session, if desired.

***ELEMLOAD** : Static Element Load Pattern

Specify an element load pattern. Repeat for each new element load pattern.

***NODALOAD** : Static Nodal Load Pattern

Specify a nodal load pattern. Repeat for each new nodal load pattern.

***ACCNREC** : Ground Acceleration Record

Specify a ground acceleration record. Repeat for each new ground acceleration record.

***SPECREC** : Response Spectrum

Specify an acceleration, velocity or displacement response spectrum. Repeat for each new response spectrum.

***DISPREC** : Ground Displacement Record

Specify a dynamic force record. Repeat for each dynamic force record.

***PARAMETERS** : Analysis Parameters

Specify control parameters for static and dynamic analyses.

***GRAV** : Static Gravity Analysis

***STAT** : Static Analysis

***REST** : Restore to Static State

***MODE** : Mode Shapes and Periods

***SPEC** : Response Spectrum Analysis

***ACCN** : New Ground Acceleration Analysis

***ACCR** : Resume Ground Acceleration Analysis

***VELN** : New Initial Velocity Analysis

***VELR** : Resume Initial Velocity Analysis

***DISN** : New Ground Displacement Analysis

***DISR** : Resume Ground Displacement Analysis

***FORN** : New Dynamic Force Analysis

***FORR** : Resume Dynamic Force Analysis

***STOP** : Stop Analysis Session

*START/*RESTART : Problem Initiation

One line.

Columns	Notes	Variable	Data
3-10(C)	S1		Problem name (1 to 8 characters).
11-15(I)	S2	NSTAT	Restart state number (*RESTART only; leave blank for *START). (a) 0 : start from the unstressed state. (b) n : start from state at the end of analysis segment "n".
20(I)	S3	KEXE	Execution code. (a) 0 : execute. (b) 1 : data checking only. (c) 2 : execute if the element data can all be held in memory, otherwise data checking only.
25(I)	S4		Input echo code. (a) 0 : do not echo. (b) 1 : echo each input line to screen as it is read.
30(I)	S5	KPDEL	P-delta analysis code. Ignored for *RESTART if NSTAT ≠ 0. (a) 0 : ignore P-delta effects. (b) 1 : consider P-delta effects.
35(I)	S6		Energy calculation code. Ignored for *RESTART if NSTAT ≠ 0. (a) 0 : omit calculations. (b) 1 : perform calculations.
41-80(C)			Problem or analysis session title.

*COMPOUND : Compound Node Types

NOTE : Compound nodes have been included for use in future versions. They have not been implemented fully in this version. Do not include this separator.

Omit for *RESTART. Omit if there are no compound nodes. See note S1 for compound node features.

Each *COMPOUND separator begins a new compound node type. Repeat the *COMPOUND separator for each compound node type.

In later sections, compound node types are referred to by node type number. Node types are numbered in the order of input.

One set of lines for each compound node type. Each line defines one subnode.

Columns	Notes	Variable	Data
1-10(R)			X offset of subnode from main node.
11-20(R)			Y offset of subnode from main node.
21-30(R)			Z offset of subnode from main node.
35(I)			Restraint code for degree of freedom 1. (a) 0 : absolute X translation, unrestrained. (b) 1 : restrained (not a degree of freedom). (c) 2 : relative X translation, unrestrained. (d) 3 : special degree of freedom, unrestrained.
36(I)			Restraint code for degree of freedom 2 (Y translation, unless a special degree of freedom).
37(I)			Restraint code for degree of freedom 3 (Z translation, unless a special degree of freedom).
38(I)			Restraint code for degree of freedom 4 (X rotation, unless a special degree of freedom).
39(I)			Restraint code for degree of freedom 5 (Y rotation, unless a special degree of freedom).
40(I)			Restraint code for degree of freedom 6 (Z rotation, unless a special degree of freedom).

*NODECOORDS : Node Coordinates

Omit for *RESTART.

"C", "L", "F", and "G" lines may be mixed in any order. As many lines as needed.

"C" Lines : Control Nodes

Columns	Notes	Variable	Data
1(C)	S1		"C", to indicate a control node.
2-10(I)			Node number.
11-20(R)			X coordinate or offset.
21-30(R)			Y coordinate or offset.
31-40(R)			Z coordinate or offset.
41-50(I)			Offset node number. Default = absolute coordinates; -1 = preceding "C" node.

"L" Lines : Straight Line Generation

Columns	Notes	Variable	Data
1(C)	S2		"L", to indicate linear generation.
2-10(I)		NB	Node number at beginning of generation line.
11-20(I)		NE	Node number at end of generation line.
21-30(I)		ND	Node number difference between successive nodes. Default = 1.
31-35(I)		NG	Number of nodes to be generated. Default = (NE-NB-1)/ND.
36-45(R)		SP	Spacing between nodes. (a) 0.0 : equally spaced along generation line. (b) < 1.0 : proportion of generation line length. (c) > 1.0 : actual spacing.

Continued

*NODECOORDS : Continued

"F" Lines : Frontal Extrapolation

Columns	Notes	Variable	Data
1(C)	S3		"F", to indicate frontal extrapolation.
2-10(I)		NB	Node number at grid origin.
11-20(I)		NE1	Node number at end of grid 1-axis.
21-30(I)		ND1	Node number difference between successive nodes along 1-axis. Default = NE1 - NB.
31-40(I)		NE2	Node number at end of grid 2-axis.
41-50(I)		ND2	Node number difference between successive nodes along 2-axis. Default = NE2 - NB.

"G" Lines : Grid Interpolation

Columns	Notes	Variable	Data
1(C)	S4		"G", to indicate grid interpolation.
2-10(I)		N1	Node number at corner 1 of grid.
11-20(I)		N2	Node number at corner 2 of grid.
21-30(I)		N3	Node number at corner 3 of grid.
31-40(I)		N4	Node number at corner 4 of grid.
41-50(I)		ND12	Node number difference between successive nodes along 1-2 axis.
51-60(I)		ND14	Node number difference between successive nodes along 1-4 axis.

*NODETYPES : Node Type Specification

Omit for *RESTART. Omit if there are no compound nodes.

As many lines as needed to specify node type numbers for all nodes. "S" and "G" lines may be mixed in any order. If a node is specified more than once, then the last specification prevails.

Default node type number = 0, i.e. a node with no subnodes.

"S" Lines : Sequential Generation

Columns	Notes	Variable	Data
1(C)	G1		"S", to indicate sequential generation.
6-10(I)	S1	NT	Node type number.
11-20(I)		NF	First node.
21-30(I)		NL	Last node. Default = NF.
31-40(I)		ND	Node number difference between successive nodes. Default = all intermediate nodes between NF and NL.

"G" Lines : Grid Generation

Columns	Notes	Variable	Data
1(C)	G2		"G", to indicate grid generation.
6-10(I)	S1	NT	Node type number.
11-20(I)		NB	Node number at grid origin.
21-30(I)		NE1	Node number at end of grid 1-axis. Default = NB.
31-40(I)		ND1	Node number difference between successive nodes along grid 1-axis. Default = NE1 - NB.
41-50(I)		NE2	Node number at end of grid 2-axis. Default = NB.
51-60(I)		ND2	Node number difference between successive nodes along grid 2-axis. Default = NE2 - NB.

*RESTRAINTS : Restrained Nodes

Omit for *RESTART. Omit if there are no nodal restraints.

As many lines as needed to specify restraint codes for all nodes. "S" and "G" lines may be mixed in any order. If a node is specified more than once, then the last specification prevails.

Default restraint code = 0, i.e. free (no restraint). For compound nodes, the restraints apply to the main node.

"S" Lines : Sequential Generation

Columns	Notes	Variable	Data
1(C)	G1		"S", to indicate sequential specification.
5(I)	S1		Restraint code for X translation. (a) 0 : free, i.e. no restraint. (b) 1 : fixed. (c) 2 : spring supported (see note). (d) 3 : no change.
6(I)			Restraint code for Y translation.
7(I)			Restraint code for Z translation.
8(I)			Restraint code for X rotation.
9(I)			Restraint code for Y rotation.
10(I)			Restraint code for Z rotation.
11-20(I)		NF	First node.
21-30(I)		NL	Last node. Default = NF.
31-40(I)		ND	Node number difference between successive nodes. Default = all intermediate nodes between NF and NL.

Continued

*RESTRAINTS : Continued

"G" Lines : Grid Generation

Columns	Notes	Variable	Data
1(C)	G2		"G", to indicate grid specification.
5(I)	S1		Restraint code for X translation. (a) 0 : free, i.e. no restraint. (b) 1 : fixed. (c) 2 : spring supported (see note). (d) 3 : no change.
6(I)			Restraint code for Y translation.
7(I)			Restraint code for Z translation.
8(I)			Restraint code for X rotation.
9(I)			Restraint code for Y rotation.
10(I)			Restraint code for Z rotation.
11-20(I)		NB	Node number at grid origin.
21-30(I)		NE1	Node number at end of grid 1-axis. Default = NB.
31-40(I)		ND1	Node number difference between successive nodes along grid 1-axis. Default = NE1 - NB.
41-50(I)		NE2	Node number at end of grid 2-axis. Default = NB.
51-60(I)		ND2	Node number difference between successive nodes along grid 2-axis. Default = NE2 - NB.

*SLAVING : Nodal Slaving Constraints

Omit for *RESTART. Omit if there are no nodal slaving constraints.

As many lines as needed to specify all nodal slaving constraints. "S" and "G" lines may be mixed in any order. A displacement once slaved can not be unslaved.

The displacements to be slaved must be free (i.e. unrestrained). The master node displacements can not themselves be slaved (i.e. only "single level" slaving is permitted). For compound nodes, the slaving constraints apply to the main node.

"S" Lines : Sequential Generation

Columns	Notes	Variable	Data
1(C)	G1		"S", to indicate sequential generation.
5(I)	S1		Slaving code for X translation. (a) 0 : no change. (b) 1 : slave.
6(I)			Slaving code for Y translation.
7(I)			Slaving code for Z translation.
8(I)			Slaving code for X rotation.
9(I)			Slaving code for Y rotation.
10(I)			Slaving code for Z rotation.
11-20(I)			Master node number.
21-30(I)		NF	First slaved node.
31-40(I)		NL	Last slaved node. Default = NF.
41-50(I)		ND	Node number difference between successive slaved nodes. Default = all intermediate nodes between NF and NL.

Continued

*SLAVING : Continued

"G" Lines : Grid Generation

Columns	Notes	Variable	Data
1(C)	G2		"G", to indicate grid generation.
5(I)	S1		Slaving code for X translation. (a) 0 : no change. (b) 1 : slave.
6(I)			Slaving code for Y translation.
7(I)			Slaving code for Z translation.
8(I)			Slaving code for X rotation.
9(I)			Slaving code for Y rotation.
10(I)			Slaving code for Z rotation.
11-20(I)			Master node number.
21-30(I)		NB	Slaved node number at grid origin.
31-40(I)		NE1	Slaved node number at end of grid 1-axis. Default = NB.
41-50(I)		ND1	Node number difference between successive slaved nodes along grid 1-axis. Default = NE1 - NB.
51-60(I)		NE2	Slaved node number at end of grid 2-axis. Default = NB.
61-70(I)		ND2	Node number difference between successive slaved nodes along grid 2-axis. Default = NE2 - NB.

*MASSES : Nodal Masses

Omit for *RESTART. Omit if there are no nodal masses.

As many lines as needed to specify all nodal masses. "S" and "G" lines may be mixed in any order. If a node is specified more than once in these lines, then its mass is the sum of the masses specified on separate lines.

"S" Lines : Sequential Generation

Columns	Notes	Variable	Data
1(C)	G1		"S", to indicate sequential generation.
3(I)	S1		Code for X mass. (a) 0 : no mass. (b) 1 : translational. (c) 2 : rotational.
4(I)			Code for Y mass.
5(I)			Code for Z mass.
6-15(R)			Mass value.
16-25(I)		NF	First node.
26-35(I)		NL	Last node. Default = NF.
36-45(I)		ND	Node number difference between successive nodes. Default = all intermediate nodes between NF and NL.
46-65(I)			Not used.
66-70(R)			Modifying factor by which mass values are to be divided. Typically "g", to convert from weight units to mass units. Default as follows : (a) First line : 1.0. (b) Subsequent lines : same as preceding line.
71-80(R)	S2		Mass proportional damping factor (α) for these masses. Default as follows : (a) First line : 0.0. (b) Subsequent lines : same as preceding line.

Continued

*MASSES : Continued

"G" Lines : Grid Generation

Columns	Notes	Variable	Data
1(C)	G1		"G", to indicate grid generation.
3(I)	S1		Code for X mass. (a) 0 : no mass. (b) 1 : translational. (c) 2 : rotational.
4(I)			Code for Y mass.
5(I)			Code for Z mass.
6-15(R)			Mass value.
16-25(I)		NB	Node number at grid origin.
26-35(I)		NE1	Node number at end of grid 1-axis. Default = NB.
36-45(I)		ND1	Node number difference between successive nodes along grid 1-axis. Default = NE1 - NB.
46-55(I)		NE2	Node number at end of grid 2-axis. Default = NB.
56-65(I)		ND2	Node number difference between successive nodes along grid 2-axis. Default = NE2 - NB.
66-70(R)			Modifying factor by which mass values are to be divided. Typically "g", to convert from weight units to mass units. Default as follows : (a) First line : 1.0. (b) Subsequent lines : same as preceding line.
71-80(R)	S2		Mass proportional damping factor (α) for these masses. Default as follows : (a) First line : 0.0. (b) Subsequent lines : same as preceding line.

*ELEMENTGROUP : Element Group Definition

Omit for *RESTART.

Each *ELEMENTGROUP separator begins a new element group (see note S1). Repeat the *ELEMENTGROUP separator for each element group.

Element groups are numbered in the order of input.

One set of lines per group.

(a) Group Information.

One line.

Columns	Notes	Variable	Data
1-5(I)	S2		Element type number.
10(I)	S3	KEVE	Event calculation code. (a) 0 : suppress event factor calculation. (b) 1 : calculate event factors for elements in this group.
15(I)	S4	KGEM	P-delta analysis code. (a) 0 : ignore P-delta effects. (b) 1 : consider P-delta effects; allow geometric stiffness to change for static analyses. (c) 2 : consider P-delta effects; allow geometric stiffness to change for both static and dynamic analyses (not recommended - see note).
21-30(R)	S5		Initial stiffness damping factor (β).
41-80(C)			Group title.

(b) Element Data.

See separate Element User Guides for details.

*SECTION : Structure Section Definition

Omit for *RESTART. Omit if there are no structure sections. See note S1 for structure section explanation.

Each *SECTION separator begins a new structure section definition. Repeat the *SECTION separator for each structure section.

Sections are numbered in the order of input.

One set of lines per structure section.

(a1) Section Title.

One line. This line may be followed by a line defining a section orientation.

Columns	Notes	Variable	Data
5(I)		KODO	Section orientation code. (a) 0 : section orientation is input on the next line. (b) 1 : section 1,2, & 3 axes ≡ global X,Y, & Z axes. (c) 2 : section 1,2, & 3 axes ≡ global Y,Z, & X axes. (d) 3 : section 1,2, & 3 axes ≡ global Z,X, & Y axes. (e) 4 : same as preceding section.
41-80(C)			Section title.

(a2) Section Orientation.

Include only if KODO = 0.

One line.

Columns	Notes	Variable	Data
1-10(R)			Projection of section 2-axis on global X-axis (in effect the direction cosine, but not necessarily normalized).
11-20(R)			Projection of section 2-axis on global Y-axis.
21-30(R)			Projection of section 2-axis on global Z-axis.
31-40(R)			Projection of section 3-axis on global X-axis.
41-50(R)			Projection of section 3-axis on global Y-axis.
51-60(R)			Projection of section 3-axis on global Z-axis.

Continued

(b1) Cut Elements.

One line for each element cut by the section. Each line may be followed by lines defining a force transformation.

Columns	Notes	Variable	Data
1-5(I)			Element group number.
6-10(I)			Element number in group.
11-20(R)			Distance along 2-axis from section center to element cut point.
21-30(R)			Distance along 3-axis from section center to element cut point.
35(I)		KODF	Force transformation code. (a) 0 : transformation is input on the immediately following lines. (b) 1 : use same transformation as for preceding cut element. (c) 2 : use same transformation as for corresponding cut element of preceding section.

(b2) Force Transformation.

Include only if KODF = 1.

NDOF lines, where NDOF = no. of dofs for cut element. Each line defines a column of the force transformation matrix.

Columns	Notes	Variable	Data
1-10(R)	S2		Transformation coefficient for X force.
11-20(R)			Transformation coefficient for Y force.
21-30(R)			Transformation coefficient for Z force.
31-40(R)			Transformation coefficient for X moment.
41-50(R)			Transformation coefficient for Y moment.
51-60(R)			Transformation coefficient for Z moment.

*GENDISP : Generalized Displacement Definition

Omit for *RESTART. Omit if there are no generalized displacements.

Each *GENDISP separator begins a new generalized displacement definition (see note S1). Repeat the *GENDISP separator for each generalized displacement.

Generalized displacements are numbered in the order of input.

One set of lines per generalized displacement, max. 8 lines per set.

Columns	Notes	Variable	Data
1-10(I)			Node number.
15(I)			Displacement direction. (a) 1 : X translation. (b) 2 : Y translation. (c) 3 : Z translation. (d) 4 : X rotation. (e) 5 : Y rotation. (f) 6 : Z rotation.
16-25(R)			Participation factor for the displacement.
41-80(C)			Generalized displacement title. Include only for the first displacement making up the set.

*RESULTS : Results Output Specification

Optional. May be included in any analysis session, whether *START or *RESTART.

If not specified for *START, the output code = 1 (i.e. post processing only, no printout) applies for results of all nodes, elements, structure sections, and generalized displacements.

If not specified for *RESTART, the specification from the preceding analysis session applies.

If an output code is specified more than once for any node, element, etc., then the last specification prevails (see note S1).

(a) Nodal Displacements.

Optional.

As many lines as needed. "S" and "G" lines may be mixed in any order. If a node is specified more than once, then the last specification prevails.

"S" Lines : Sequential Generation

Columns	Notes	Variable	Data
1-2(C)	G1		"NS", to indicate nodal results and sequential generation.
5(I)	S1		Output code. (a) 0 : neither post-processing nor printout. (b) 1 : post-processing only, no printout. (c) 2 : both post-processing and printout.
11-20(I)	S2	NF	First node. Default = all nodes. NL and ND are ignored.
21-30(I)		NL	Last node. Default = NF.
31-40(I)		ND	Node number difference between successive nodes. Default = all intermediate nodes between NF and NL.

Continued

*RESULTS : Continued

"G" Lines : Grid Generation

Columns	Notes	Variable	Data
1-2(C)	G2		"NG", to indicate nodal results and grid generation.
5(I)	S1		Output code. (a) 0 : neither post-processing nor printout. (b) 1 : post-processing only, no printout. (c) 2 : both post-processing and printout.
11-20(I)	S2	NB	Node number at grid origin.
21-30(I)		NE1	Node number at end of grid 1-axis. Default = NB.
31-40(I)		ND1	Node number difference between successive nodes along grid 1-axis. Default = NE1 - NB.
41-50(I)		NE2	Node number at end of grid 2-axis. Default = NB.
51-60(I)		ND2	Node number difference between successive nodes along grid 2-axis. Default = NE2 - NB.

Continued

*RESULTS : Continued

(b) Elements.

Optional.

As many lines as needed. If an element is specified more than once, then the last specification prevails.

Columns	Notes	Variable	Data
1(C)			"E", to indicate element results.
5(I)			Output code. (a) 0 : neither post-processing nor printout. (b) 1 : post-processing only, no printout. (c) 2 : both post-processing and printout.
6-10(I)			Element Group Number. Default = all elements in all groups. NF, NL and ND are ignored.
11-15(I)		NF	First element number. Default = all elements in this group. NL and ND are ignored.
16-20(I)		NL	Last element number. Default = NF.
21-25(I)		ND	Element number difference between successive elements. Default = 1.

Continued

*RESULTS : Continued

(c) Structure Sections.

Omit if there are no structural sections.

Optional.

As many lines as needed. If a section is specified more than once, then the last specification prevails.

Columns	Notes	Variable	Data
1(C)			"S", to indicate structure section results.
5(I)			Output code. (a) 0 : neither post-processing nor printout. (b) 1 : post-processing only, no printout. (c) 2 : both post-processing and printout.
6-10(I)		NF	First section number. Default = all sections. NL and ND are ignored.
11-15(I)		NL	Last section number. Default = NF.
16-20(I)		ND	Section number difference between successive sections. Default = 1.

Continued

*RESULTS : Continued

(d) Generalized Displacements.

Omit if there are no generalized displacements.

Optional.

As many lines as needed. If a generalized displacement is specified more than once, then the last specification prevails.

Columns	Notes	Variable	Data
1(C)			"G", to indicate generalized displacement results.
5(I)			Output code. (a) 0 : neither post-processing nor printout. (b) 1 : post-processing only, no printout. (c) 2 : both post-processing and printout.
6-10(I)		NF	First generalized displacement number. Default = all generalized displacements. NL and ND are ignored.
11-15(I)		NL	Last generalized displacement number. Default = NF.
16-20(I)		ND	Generalized displacement number difference between successive generalized displacements. Default = 1.

*ELEMLOAD : Static Element Load Pattern

Optional. May be included with *START or *RESTART. If any element load patterns have already been defined, the new patterns are added.

Each *ELEMLOAD separator begins a new element load pattern. See note S1.

One set of lines per pattern. Each set consists of a pattern name plus one subset of lines for each affected element group. The data needed to define the loads for an element group depends on the element type.

Note that element load patterns can be used only for gravity load analysis, and not for general static analysis.

(a) Pattern Name.

One line.

Columns	Notes	Variable	Data
2-5(C)	G3		Pattern name.
41-80(C)			Pattern title.

(b1) Element Group.

One line, to begin an element group. Repeat this line for each affected group. The groups must be in increasing numerical order.

Columns	Notes	Variable	Data
1(C)			"G", to indicate beginning of an affected element group.
2-5(I)	S2		Element group number.
6-10(I)	S3	NLOD	Number of element load sets (max. 40) for this group.

(b2) Element Loads.

The element load data depends on the element type. See separate Element User Guides for details.

*NODALOAD : Static Nodal Load Patterns

Optional. May be included with *START or *RESTART. If any nodal load patterns have already been defined, the new patterns are added.

Each *NODALOAD separator begins a new nodal load pattern.

One set of lines per pattern. Each set consists of a pattern name plus as many lines as required to specify the nodal loads.

(a) Pattern Name.

One line.

Columns	Notes	Variable	Data
2-5(C)	G3		Pattern name.
41-80(C)			Pattern title.

(b) Nodal Loads.

As many lines as needed. "S" and "G" lines may be mixed in any order. If a node is specified more than once in these lines, then the last specification prevails.

"S" Lines : Sequential Generation

Columns	Notes	Variable	Data
1(C)	G1		"S", to indicate sequential generation.
2(C)			Code for load type. (a) F : forces. (b) M : moments.
3-10(R)			X load.
11-20(R)			Y load.
21-30(R)			Z load.
31-40(I)		NF	First node.
41-50(I)		NL	Last node. Default = NF.
51-60(I)		ND	Node number difference between successive nodes. Default = all intermediate nodes between NF and NL.

Continued

*NODALOADS : Continued

"G" Lines : Grid Generation

Columns	Notes	Variable	Data
1(C)	G2		"G", to indicate grid generation.
2(C)			Code for load type. (a) F : forces. (b) M : moments.
3-10(R)			X load.
11-20(R)			Y load.
21-30(R)			Z load.
31-40(I)		NB	Node number at grid origin.
41-50(I)		NE1	Node number at end of grid 1-axis. Default = NB.
51-60(I)		ND1	Node number difference between successive nodes along grid 1-axis. Default = NE1 - NB.
61-70(I)		NE2	Node number at end of grid 2-axis. Default = NB.
71-80(I)		ND2	Node number difference between successive nodes along grid 2-axis. Default = NE2 - NB.

*ACCNREC : Ground Acceleration Record

Optional. May be included with *START or *RESTART. If any acceleration records have already been defined, the new records are added.

Each *ACCNREC separator begins a new ground acceleration record.

One set of lines per record.

(a) Record Data.

One line.

Columns	Notes	Variable	Data
2-5(C)	G3		Record name.
6-20(C)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.
21-40(C)			Data format. Eg. (8f10.0) ; (4(f8.5,f12.5)) ; etc. Leave blank for reading from unformatted file. Use (*) for reading in free format.
41-80(C)			Record title.

Continued

*ACCNREC : Continued

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-5(I)		NVAL	Total number of acceleration values to be read.
6-10(I)		NVLIN	Number of acceleration values per input data line (max. 20).
15(I)		KODE	Data code. (a) 0 : acceleration values only, assumed at constant interval DT. (b) 1 : time-acceleration pairs. (c) 2 : acceleration-time pairs.
20(I)			Print code. (a) 0 : print as input (b) 1 : print as scaled (c) 2 : no print.
21-30(R)	S1		Time scale factor. Default = 1.0. All input times are multiplied by this factor.
31-40(R)	S2		Acceleration scale factor. Default = 1.0. All input accelerations are multiplied by this factor.
41-50(R)		DT	Time interval if KODE = 0. Default = 1.0. Ignored if KODE ≠ 0.
51-60(R)			Start time if KODE = 0. Ignored if KODE ≠ 0.

(c) Acceleration Values.

As many lines as needed to specify NVAL acceleration values in the format and on the file specified in (a). Each line should have NVLIN (max. 20) acceleration values. The maximum line length is 160 characters for formatted or free format data.

Columns	Notes	Variable	Data
			Time-acceleration pairs or acceleration values.

*SPECTRUM : Response Spectrum

Optional. May be included with *START or *RESTART. If any spectra have already been defined, the new spectra are added.

Each *SPECTRUM separator begins a new response spectrum.

One set of lines per spectrum.

(a) Spectrum Data.

One line.

Columns	Notes	Variable	Data
2-5(C)	G3		Spectrum name.
6-20(C)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.
21-40(C)			Data format. Eg. (8f10.0) ; (4(f8.5,f12.5)) ; etc. Leave blank for reading from unformatted file. Use (*) for reading in free format.
41-80(C)			Spectrum title.

Continued

*SPECTRUM : Continued

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-5(I)		NVAL	Number of response values to be read (max. 141).
6-10(I)		NVLIN	Number of response values per input data line.
15(I)			Spectrum type code. (a) 0 : acceleration. (b) 1 : velocity. (c) 2 : displacement.
20(I)		KODE	Data code. (a) 0 : response values only, assumed at constant period intervals. (b) 1 : period-response pairs. (c) 2 : response-period pairs. (d) 3 : response values only, assumed at constant frequency intervals. (e) 4 : frequency-response pairs. (f) 5 : response-frequency pairs.
25(I)			Print code. (a) 0 : print as input (b) 1 : print as scaled (c) 2 : no print.
26-35(R)	S1		Period scale factor if KODE = 0,1 or 2. Frequency scale factor if KODE = 3,4 or 5. Default = 1.0. All input periods or frequencies are multiplied by this factor.
36-45(R)	S2		Response scale factor. Default = 1.0. All input response values are multiplied by this factor.
46-55(R)			Period interval if KODE = 0. Frequency interval if KODE = 3. Default = 1.0. Ignored if KODE ≠ 0 or 3.
56-65(R)			Starting period if KODE = 0. Starting frequency if KODE = 3. Ignored if KODE ≠ 0 or 3.

Continued

*SPECTRUM : Continued

(c) Response Values.

As many lines as needed to specify NVAL response values in the format and on the file specified in (a). Each line should have NVLIN response values. The maximum line length is 160 characters for formatted or free format data.

Columns	Notes	Variable	Data
			Time (or frequency) - response pairs or response values.

*NODALVEL : Initial Velocity Patterns

Optional. May be included with *START or *RESTART. If any initial velocity patterns have already been defined, the new patterns are added.

Each *NODALVEL separator begins a new initial velocity pattern. See note S1 for use of initial velocities.

One set of lines per pattern. Each set consists of a pattern name plus as many lines as required to specify nodal velocities.

(a) Pattern Name.

One line.

Columns	Notes	Variable	Data
2-5(C)	G3		Pattern name.
41-80(C)			Pattern title.

(b) Nodal Velocities.

As many lines as needed. "S" and "G" lines may be mixed in any order. If a node is specified more than once in these lines, then the last specification prevails.

"S" Lines : Sequential Generation

Columns	Notes	Variable	Data
1(C)	G1		"S", to indicate sequential generation.
2(C)			Code for velocity type. (a) T : translational. (b) R : rotational.
3-10(R)			X Velocity.
11-20(R)			Y Velocity.
21-30(R)			Z Velocity.
31-40(I)	S2	NF	First node.
41-50(I)		NL	Last node. Default = NF.
51-60(I)		ND	Node number difference between successive nodes. Default = all intermediate nodes between NF and NL.

Continued

*NODALVEL : Continued

"G" Lines : Grid Generation

Columns	Notes	Variable	Data
1(C)	G2		"G", to indicate grid generation.
2(C)			Code for velocity type. (a) T : translational. (b) R : rotational.
3-10(R)			X Velocity.
11-20(R)			Y Velocity.
21-30(R)			Z Velocity.
31-40(I)	S2	NB	Node number at grid origin.
41-50(I)		NE1	Node number at end of grid 1-axis. Default = NB.
51-60(I)		ND1	Node number difference between successive nodes along grid 1-axis. Default = NE1 - NB.
61-70(I)		NE2	Node number at end of grid 2-axis. Default = NB.
71-80(I)		ND2	Node number difference between successive nodes along grid 2-axis. Default = NE2 - NB.

*DISPREC : Ground Displacement Record

Optional. May be included with *START or *RESTART. If any displacement records have already been defined, the new records are added.

Each *DISPREC separator begins a new ground displacement record. See note S1 for use of displacement record.

One set of lines per record.

(a) Record Data.

One line.

Columns	Notes	Variable	Data
2-5(C)	G3		Record name.
6-20(C)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.
21-40(C)			Data format. Eg. (8f10.0) ; (4(f8.5,f12.5)) ; etc. Leave blank for reading from unformatted file. Use (*) for reading in free format.
41-80(C)			Record title.

Continued

*DISPREC : Continued

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-5(I)		NVAL	Total number of displacement values to be read.
6-10(I)		NVLIN	Number of displacement values per input data line (max. 20).
15(I)		KODE	Data code. (a) 0 : displacement values only, assumed at constant interval DT. (b) 1 : time-displacement pairs. (c) 2 : displacement-time pairs.
20(I)			Print code. (a) 0 : print as input (b) 1 : print as scaled (c) 2 : no print.
21-30(R)	S2		Time scale factor. Default = 1.0. All input times are multiplied by this factor.
31-40(R)	S3		Displacement scale factor. Default = 1.0. All input displacements are multiplied by this factor.
41-50(R)		DT	Time interval if KODE = 0. Default = 1.0. Ignored if KODE ≠ 0.
51-60(R)			Start time if KODE = 0. Ignored if KODE ≠ 0.

(c) Displacement Values.

As many lines as needed to specify NVAL displacement values in the format and on the file specified in (a). Each line should have NVLIN (max. 20) displacement values. The maximum line length is 160 characters for formatted or free format data.

Columns	Notes	Variable	Data
			Time-displacement pairs or displacement values.

*FORCREC : Dynamic Force Record

Optional. May be included with *START or *RESTART. If any force records have already been defined, the new records are added.

Each *FORCREC separator begins a new dynamic force record.

One set of lines per record.

(a) Record Data.

One line.

Columns	Notes	Variable	Data
2-5(C)	G3		Record name.
6-20(C)			Name of file (max. 12 characters) from which data is to be read. Default = read from input file.
21-40(C)			Data format. Eg. (8f10.0) ; (4(f8.5,f12.5)) ; etc. Leave blank for reading from unformatted file. Use (*) for reading in free format.
41-80(C)			Record title.

Continued

*FORCREC : Continued

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-5(I)		NVAL	Total number of force values to be read.
6-10(I)		NVLIN	Number of force values per input data line (max. 20).
15(I)		KODE	Data code. (a) 0 : force values only, assumed at constant interval DT. (b) 1 : time-force pairs. (c) 2 : force-time pairs.
20(I)			Print code. (a) 0 : print as input (b) 1 : print as scaled (c) 2 : no print.
21-30(R)			Time scale factor. Default = 1.0. All input times are multiplied by this factor.
31-40(R)			Force scale factor. Default = 1.0. All input forces are multiplied by this factor.
41-50(R)		DT	Time interval if KODE = 0. Default = 1.0. Ignored if KODE ≠ 0.
51-60(R)			Start time if KODE = 0. Ignored if KODE ≠ 0.

(c) Force Values.

As many lines as needed to specify NVAL force values in the format and on the file specified in (a). Each line should have NVLIN (max. 20) force values. The maximum line length is 160 characters for formatted or free format data.

Columns	Notes	Variable	Data
			Time-force pairs or force values.

*PARAMETERS : Analysis Parameters

Optional. May be included with *START or *RESTART. If not included for *START, the initial defaults apply. If not included for *RESTART, the specifications for the preceding analysis session apply.

These parameters can be changed before the first analysis segment in any analysis session or between analysis segments, as desired.

The analysis parameters are used to control the static and dynamic nonlinear analyses. These parameters will often be specified only once for a structure, and will be used for all subsequent analyses of that structure. However, it may be necessary to change the parameters between analysis segments.

(a) Viscous Damping Scale Factors.

Optional.

The α and β damping values may be scaled if the structure is in a static state. If the structure is not in a static state, then this line will be ignored. The scale factors are applied to the original values specified in *MASSES and *ELEMENTGROUP.

One line.

Columns	Notes	Variable	Data
1(C)			"V", to indicate viscous damping scale factors.
2-10(R)			Scale factor for α damping. Initial default = 1.0.
11-20(R)			Scale factor for β damping. Initial default = 1.0.

Continued

*PARAMETERS : Continued

(b) Collapse Displacements.

Optional.

Used to change the values of the displacements at which collapse of structure may be assumed and analysis terminated.

One line.

Columns	Notes	Variable	Data
1(C)			"C", to indicate collapse displacements.
2-10(R)			Collapse translation for static analysis. Default = no limit. If the X, Y or Z translation at any node exceeds this value, the analysis quits.
11-20(R)			Collapse rotation for static analysis. Default = no limit. If the X, Y or Z rotation at any node exceeds this value, the analysis quits.
21-30(R)			Collapse translation for dynamic analysis. Default = no limit. If the X, Y or Z translation at any node exceeds this value, the analysis quits.
31-40(R)			Collapse rotation for dynamic analysis. Default = no limit. If the X, Y or Z rotation at any node exceeds this value, the analysis quits.

Continued

(c) Event Overshoot Scale Factors.

Optional.

Used to change the event overshoot scale factors for element groups.

Overshoot tolerances are specified as part of the element data. These tolerances can be scaled for each element group by specifying overshoot scale factors. The overshoot tolerance for analysis is the value input with the element data multiplied by the overshoot scale factor. The initial default is 1.0 for all element groups.

As many lines as needed. The last specification for any element group prevails.

Columns	Notes	Variable	Data
1(C)			"F", to indicate event overshoot scale factors.
2-5(I)			Element group number. Default = all groups.
6-15(R)			Event overshoot scale factor for static analyses. Initial default = 1.0.
16-25(R)			Event overshoot scale factor for dynamic analyses. Initial default = 1.0.

Continued

*PARAMETERS : Continued

(d) Element Parameters.

Optional.

Used to change the element parameters for element groups. The initial default is zero for all element parameters for all element groups.

Provision is made for specifying a maximum of 2 integer and 2 real parameters for each element group. The use of these parameters is element-type dependent.

As many lines as needed. The last specification for any element group prevails.

Columns	Notes	Variable	Data
1(C)			"E", to indicate element parameters.
6-10(I)			Element group number. Default = all element groups.
11-15(I)			1st integer parameter.
16-20(I)			2nd integer parameter.
21-30(R)			1st real parameter.
31-40(R)			2nd real parameter.

Continued

*PARAMETERS : Continued

(e) Output Intervals for Static Analyses.

Optional.

Used to set the intervals for saving structure state, saving results for post-processing, results printout, saving envelopes for post-processing, and envelope printout for static analyses (*GRAV, *STAT, and *REST).

One line.

Columns	Notes	Variable	Data
1-2(C)			"OS", to indicate output intervals for static analyses.
6-10(I)	S1	ISAVES	Load step interval for saving structure state. (a) 0 : do not save. (b) +n : save every 'n' steps, and at end of analysis segment. Initial default = +∞.
11-15(I)	S2	IPPSVS	Load step interval for saving results for post-processing. (a) -1 : save every event. (b) 0 : do not save. (c) +n : save every 'n' steps, and at end of analysis segment. Initial default = 1.
16-20(I)	S3	IPOUTS	Load step interval for results printout. (a) -1 : print every event. (b) 0 : do not print. (c) +n : print every 'n' steps, and at end of analysis segment. Initial default = 0.
21-25(I)	S4	IENVSS	Load step interval for saving envelopes for post processing. (a) 0 : do not save. (b) +n : save every 'n' steps, and at end of analysis segment. Initial default = +∞.
26-30(I)	S5	IENVPS	Load step interval for envelope printout. (a) 0 : do not print. (b) +n : print every 'n' steps, and at end of analysis segment. Initial default = +∞.

Continued

*PARAMETERS : Continued

(f) Output Intervals for Dynamic Analysis.

Optional.

Used to set the intervals for saving structure state, saving results for post-processing, results printout, saving envelopes for post-processing, and envelope printout for dynamic analyses (*ACCN, *ACCR, *VELN, *VELR, *DISN, *DISR, *FORN, and *FORR).

One line.

Columns	Notes	Variable	Data
1-2(C)			"OD", to indicate output intervals for dynamic analysis.
6-10(I)	S1	ISAVED	Step interval for saving structure state. (a) 0 : not used. (b) +n : save every 'n' steps, unless TSAVED governs, and at end of analysis segment. Initial default = $+\infty$.
11-20(R)		TSAVED	Time interval for saving structure state. (a) 0. : not used. (b) +v : save at this time interval, unless ISAVED governs, and at end of analysis segment. Initial default = 0.0. If both ISAVED & TSAVED are zero, the state is not saved.
21-25(I)	S2	IPPSVD	Step interval for saving results for post-processing. (a) 0 : not used. (b) +n : save every 'n' steps, unless TPPSVD governs, and at end of analysis segment. Initial default = 1.
26-35(R)		TPPSVD	Time interval for saving results for post-processing. (a) 0. : not used. (b) +v : save at this time interval, unless IPPSVD governs, and at end of analysis segment. Initial default = 0.0. If both IPPSVD & TPPSVD are zero, the results are not saved.

36-40(I)	S3	IPOUTD	Step interval for results printout. (a) 0 : not used. (b) +n : print every 'n' steps, unless TPOUTD governs, and at end of analysis segment. Initial default = 0.
41-50(R)		TPOUTD	Time interval for results printout. (a) 0. : not used. (b) +v : print at this time interval, unless IPOUTD governs, and at end of analysis segment. Initial default = 0.0. If both IPOUTD & TPOUTD are zero, the results are not printed.
51-55(I)	S4	IENVSD	Time step interval for saving envelopes for post processing. (a) 0 : not used. (b) +n : save every 'n' steps, unless TENVSD governs, and at end of analysis segment. Initial default = +∞.
56-65(R)		TENVSD	Time interval for saving envelopes for post processing. (a) 0. : not used. (b) +v : save at this time interval, unless IENVPD governs, and at end of analysis segment. Initial default = 0.0. If both IENVSD & TENVSD are zero, the envelopes are not saved.
66-70(I)	S5	IENVPD	Time step interval for envelope printout. (a) 0 : not used. (b) +n : print every 'n' steps, unless TENVPD governs, and at end of analysis segment. Initial default = +∞.
71-80(R)		TENVPD	Time interval for envelope printout. (a) 0. : not used. (b) +v : print at this time interval, unless IENVPD governs, and at end of analysis segment. Initial default = 0.0. If both IENVPD & TENVPD are zero, the envelopes are not printed.

Continued

(g) Control Parameters for Dynamic Analysis.

Optional.

Used to change some of the control parameters for dynamic analyses (*ACCN, *ACCR, *VELN, *VELR, *DISN, *DISR, *FORN, and *FORR).

One line.

Columns	Notes	Variable	Data
1-2(C)			"DC", to indicate dynamic analysis control parameters.
5(I)	S8	KEVNT	Event calculation code. (a) 0 : ignore events. (b) 1 : consider events within time steps. Initial default = 0.
10(I)	S6	KENRC	Velocity correction code. Used only if KEVNT = 1. (a) 0 : do not modify velocities. (b) 1 : modify velocities (to satisfy energy balance). Initial default = 0. KENRC is ignored if KEVNT = 0.
15(I)	S6	KEQBC	Acceleration correction code. (a) 0 : do not modify accelerations. (b) 1 : modify accelerations (to improve equilibrium). Initial default = 0.
16-20(I)	S7	MAXEV	Maximum number of events allowed in a time step. (a) -n : stop checking for events if there are more than 'n' events in current step, but continue execution. (b) 0 : no limit. (c) +n : stop execution if there are more than 'n' events in any step. Initial default = 0. MAXEV is ignored if KEVNT = 0.

Continued

*PARAMETERS : Continued

(h) Time Step Parameters for Dynamic Analysis.

Optional.

Used to set the time steps for dynamic analyses (*ACCN, *ACCR, *VELN, *VELR, *DISN, *DISR, *FORN, and *FORR). See note S8.

One line.

Columns	Notes	Variable	Data
1-2(C)			"DT", to indicate dynamic analysis time step parameters.
2-10(R)			Time step for constant time step solution scheme. Initial default = 1.0.
11-20(R)			Initial time step for variable time step solution scheme. Initial default = 1.0.
21-30(R)		DTMAX	Maximum allowable time step for variable time step solution scheme. Default = no limit.
31-40(R)		DTMIN	Minimum allowable time step for variable time step solution scheme. Default = 0.0.

Continued

*PARAMETERS : Continued

(i) Parameters for Variable Time Step Solution Scheme.

Optional.

Used to set the control parameters for the variable time step (KAUTO=2 option) solution scheme for dynamic analyses (*ACCN, *ACCR, *VELN, *VELR, *DISN, *DISR, *FORN, and *FORR). See note S8.

One line.

Columns	Notes	Variable	Data
1-2(C)			"DA", to indicate dynamic analysis variable time step parameters.
6-15(R)		TOLHIS	Upper static force error tolerance. Default = no limit.
16-25(R)		TOLHII	Upper impulse force error tolerance. Default = no limit.
26-35(R)		DTRED	Time step decrease factor (TOLHIS or TOLHII exceeded). Default = 0.5. Time step is never decreased below DTMIN (see DT line of *PARAMETERS).
36-45(R)		TOLLOS	Lower static force error tolerance. Default = 0.9 * TOLHIS * DTRED * DTRED.
46-55(R)		TOLLOI	Lower impulse force error tolerance. Default = 0.9 * TOLHII * DTRED * DTRED.
56-60(I)		NSINC	Number of steps below TOLLOS and TOLLOI before time step is increased. Default = 2.
61-70(R)		DTINC	Time step increase factor. Default = 1/DTRED. Time step is never increased above DTMAX (see DT line of *PARAMETERS).
71-80(R)		TOLMX	Maximum force error tolerance. Analysis quits if exceeded. Default = no limit.

*GRAV : Static Gravity Analysis

If a gravity analysis is performed, it must begin with the unstressed state. Also the behavior under gravity loads must be linear.

(a) Analysis Title.

Place title in columns 41-80 of *GRAV line.

(b) Element Loads.

As many lines as needed to specify element loads, one element pattern per line.

Columns	Notes	Variable	Data
1(C)			"E", to indicate element loads.
7-10(C)			Element load pattern name (see *ELEMLOAD).
11-20(R)			Pattern scale factor. Default = 1.0

(c) Nodal Loads.

As many lines as needed to specify nodal loads, one nodal load pattern per line.

Columns	Notes	Variable	Data
1(C)			"N", to indicate nodal loads.
7-10(C)			Nodal load pattern name (see *NODALOAD).
11-20(R)			Pattern scale factor. Default = 1.0

Continued

(d) Inertial Loads.

Optional.

One line.

Columns	Notes	Variable	Data
1(C)			"I", to indicate inertial loads.
2-10(R)			Modifying factor by which mass values are to be multiplied. Typically "g", to convert from mass units to weight units. Default = 1.0.
11-20(R)			X direction scale factor. Typically 0., if X direction is horizontal.
21-30(R)			Y direction scale factor. Typically 0., if Y direction is horizontal.
31-40(R)			Z direction scale factor. Typically -1., if Z direction is vertical and postive upwards.

*STAT : Static Analysis

The structure must be in static state at the start of this analysis (the preceding analysis segment, if any, must have been *REST or *STAT or *GRAV).

The loading for this analysis segment will usually be applied in a number of steps, controlled by load and/or displacement increments. Within each step, an event-to-event solution strategy is used, dividing each step into substeps at each event.

Loads corresponding to a load factor of unity are first defined (section (b) and (c)). The same loads as for the preceding (*STAT) static analysis can be specified if desired.

The analysis proceeds under load control (section (d)) and/or under displacement control (section (e)). The analysis segment is complete when, either the load factor increment (under load control) or the displacement increment (under displacement control) for the segment is reached.

To avoid waste of computing time if the solution flip-flops, additional limits may be placed on the number of events in any step, maximum number of steps (useful only if both load control and displacement control are specified), and on the number of successive direction changes (under displacement control only). The analysis quits if these values are exceeded.

(a) Analysis Title.

Place title in columns 41-80 of *STAT line.

(b) Nodal Loads.

As many lines as needed to specify nodal loads corresponding to unit load factor, one nodal load pattern per line.

Columns	Notes	Variable	Data
1(C)			"N", to indicate nodal loads.
7-10(C)	S1		Nodal load pattern name (see *NODALOAD). Leave first line blank, to use loads from the preceding static analysis - see note for meaning.
11-20(R)			Pattern scale factor. Default = 1.0.

Continued

*STAT : Continued

(c) Inertial Loads.

Optional.

One line.

Columns	Notes	Variable	Data
1(C)			"I", to indicate inertial loads.
2-10(R)			Modifying factor by which mass values are to be multiplied. Typically "g", to convert from mass units to weight units. Default = 1.0.
11-20(R)			X direction scale factor. Typically 0., if X direction is horizontal.
21-30(R)			Y direction scale factor. Typically 0., if Y direction is horizontal.
31-40(R)			Z direction scale factor. Typically -1., if Z direction is vertical and postive upwards.

(d) Load Control.

Optional if displacement control is to be specified (section (e)).

One line.

Columns	Notes	Variable	Data
1(C)			"L", to indicate load control.
2-10(R)	S2	STPLF	Load factor increment for each analysis step. Must be positive. Default = 1.0.
11-20(R)		SEGLF	Load factor increment for analysis segment. Default = 1.0.
21-25(I)		MAXEV	Maximum number of events in any load step (analysis quits if exceeded). Default = no limit.

Continued

(e) Displacement Control.

Optional if load control is specified (section (d)).

If a displacement control is specified, the increment applies to a specified "controlled displacement", which is equal to the 1st node displacement minus the 2nd node displacement.

One line.

Columns	Notes	Variable	Data
1(C)	S3		"D", for displacement control.
2-10(I)			Node number for 1st node. Default = use minus 2nd node displacement.
11-20(I)			Node number of 2nd node. Default = use 1st node displacement.
25(I)			Displacement direction code. (a) 1 : X translation. (b) 2 : Y translation. (c) 3 : Z translation. (d) 4 : X rotation. (e) 5 : Y rotation. (f) 6 : Z rotation. Default = 1.
26-35(R)		STPDI	Displacement increment per step. Must be positive. Default as follows : (a) with load control : no limit. (b) without load control : none. Load factor magnitude and sign is chosen so that controlled displacement increases by this amount in each step. However, if load control is also specified (section (d)), then the load factor magnitude will not exceed the value specified there.
36-45(R)		SEGDI	Displacement increment for analysis segment. Default as follows : (a) with load control : no limit. (b) without load control : none.

46-50(I)		MAXEV	Maximum number of events in any step (analysis quits if exceeded). Default as follows : (a) with load control : as in load control. (b) without load control : no limit.
51-55(I)	S4	MAXFP	Maximum number of successive direction changes for load increment (flip-flop control - analysis quits if exceeded). Default = 3.
56-60(I)	S5	NSTEPS	Maximum number of steps (analysis quits if exceeded). Omit if load control is not specified. Default = 1 + Maximum of SEGLF/STPLF & SEGDI/STPDI.

*REST : Restore to Static State

The preceding analysis segment must have been a dynamic analysis (*ACCN, *ACCR, *VELN, *VELR, *DISN, *DISR, *FORN, *FORR). The inertia and damping forces acting at end of the preceding segment are applied as static loads, in order to bring the structure to a static equilibrium state.

(a) Analysis Title.

Place title in columns 41-80 of *REST line.

There is no other input data for this analysis segment.

*MODE : Mode Shapes and Periods

Mode shapes and periods may be calculated for any state of the structure. The preceding analysis can be either static or dynamic.

A *MODE analysis is not an analysis segment.

If the save option is selected, then a .MXX file will be created for post-processing and for response spectrum analysis, where M indicates a mode shape file and XX is the number of the preceding analysis segment.

(a) Analysis Title.

Place title in columns 41-80 of *MODE line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-5(I)		NVEC	Maximum number of mode shapes to be calculated. Default = controlled by TMIN.
6-15(R)		TMIN	Shortest significant period. Mode shapes with periods < TMIN are not calculated. Default = controlled by NVEC.
20(I)			Code for printing and saving mode shapes. (a) 0 : print and save. (b) 1 : print only. (c) 2 : save only.

*SPEC : Response Spectrum Analysis

This analysis can be used to perform a linear response spectrum analysis. As a general rule, only mode shapes and frequencies calculated for either (a) the unstressed state, or (b) the state under gravity loads should be used. The results are likely to be meaningless for mode shapes based on other states. The mode shapes and frequencies must have been calculated and saved prior to this analysis segment. Square root of sum of squares (SRSS) combination is used to combine the modal responses.

A *SPEC analysis is not an analysis segment.

(a) Analysis Title.

Place title in columns 41-80 of *SPEC line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-5(I)		KSEG	Analysis segment number for which mode shapes and periods were calculated.
10(I)			Print code. (a) 0 : results for all nodal displacements, elements, sections, and generalized displacements. (b) 1 : results as for post-processing (see the *RESULTS separator). (c) 2 : results as for printout (see the *RESULTS separator).
17-20(C)			Name of X direction spectrum (see *SPECTRUM). Default = none.
21-30(R)			X spectrum scale factor. Default = 1.0.
32-35(C)			Name of Y direction spectrum. Default = none.
36-45(R)			Y spectrum scale factor. Default = 1.0.
47-50(C)			Name of Z direction spectrum. Default = none.
51-60(R)			Z spectrum scale factor. Default = 1.0.

*ACCN : New Ground Acceleration Analysis

The structure must be in static state at the start of this analysis (the preceding analysis segment, if any, must have been *REST or *STAT or *GRAV).

This analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

(a) Analysis Title.

Place title in columns 41-80 of *ACCN line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Segment starts at time zero and ends at time SEGTM.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 1 : constant time step. (b) 2 : variable time step.
21-30(R)		DT	Optional time step, as follows : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.
31-40(R)			X coordinate of center of rotation. Used only if rotational accelerations are imposed.
41-50(R)			Y coordinate of center of rotation. Used only if rotational accelerations are imposed.
51-60(R)			Z coordinate of center of rotation. Used only if rotational accelerations are imposed.

Continued

(c) Ground Acceleration Records.

As many lines as needed to specify the ground accelerations (max. 6 lines).

Columns	Notes	Variable	Data
1(I)		KDIR	Acceleration direction code. (a) 1 : X translation. (b) 2 : Y translation. (c) 3 : Z translation. (d) 4 : X rotation. (e) 5 : Y rotation. (f) 6 : Z rotation.
7-10(C)			Name of acceleration record (see *ACCNREC).
11-20(R)			Acceleration scale factor. Default = 1.0.
21-30(R)			Time scale factor. Default = 1.0.

*ACCR : Resume Ground Acceleration Analysis

The preceding analysis segment must have been *ACCN or *ACCR. The same ground acceleration records, as for the preceding *ACCN or *ACCR analysis are used.

(a) Analysis Title.

Place title in columns 41-80 of *ACCR line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Time at the end of segment will be time at beginning plus this increment.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 0 : continue with same KAUTO and same time step as preceding analysis segment. (b) 1 : change to constant time step. (c) 2 : change to variable time step.
21-30(R)		DT	Optional time step, as follows (KAUTO = 1 or 2 only) : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.

*VELN : New Initial Velocity Analysis

The structure must be in static state at the start of this analysis (the preceding analysis segment, if any, must have been *REST or *STAT or *GRAV).

This analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

(a) Analysis Title.

Place title in columns 41-80 of *VELN line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Segment starts at time zero and ends at time SEGTM.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 1 : constant time step. (b) 2 : variable time step.
21-30(R)		DT	Optional time step, as follows : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.
31-40(R)		EKIN	Initial kinetic energy (optional). Default = patterns as specified in section (c) are used directly. If EKIN > 0.0, the patterns are combined, then all velocities are scaled to give the specified initial kinetic energy.

(c) Initial Velocity Patterns.

As many lines as needed to specify initial nodal velocities, one velocity pattern per line.

Columns	Notes	Variable	Data
2-5(C)			Initial velocity pattern name (see *NODALVEL).
6-15(R)			Pattern scale factor. Default = 1.0.

*VELR : Resume Initial Velocity Analysis

The preceding analysis segment must have been *VELN or *VELR. The initial velocities at the end of the preceding analysis are used.

(a) Analysis Title.

Place title in columns 41-80 of *VELR line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Time at the end of segment will be time at beginning plus this increment.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 0 : continue with same KAUTO and same time step as preceding analysis segment. (b) 1 : change to constant time step. (c) 2 : change to variable time step.
21-30(R)		DT	Optional time step, as follows (KAUTO = 1 or 2 only) : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.

*DISN : New Ground Displacement Analysis

The structure must be in static state at the start of this analysis (the preceding analysis segment, if any, must have been *REST or *STAT or *GRAV).

This analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

(a) Analysis Title.

Place title in columns 41-80 of *DISN line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Segment starts at time zero and ends at time SEGTM.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 1 : constant time step. (b) 2 : variable time step.
21-30(R)		DT	Optional time step, as follows : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.

Continued

(c) Ground Displacement Records.

One line per applied ground displacement record (max. 36). Each record may be associated with one or more nodal degrees of freedom in (d). To represent a ground wave, use the same record with different time delays. The applied records are numbered in the order of input.

Columns	Notes	Variable	Data
1(C)			"R", to indicate ground displacement record.
7-10(C)			Record name (see *DISPREC).
11-20(R)			Time scale factor. Default = 1.0.
21-30(R)			Displacement scale factor for the record. Default = 1.0.
31-40(R)		DTIME	Time delay.

Continued

*DISN : Continued

(d) Degrees of Freedom.

These degrees of freedom must be spring supported (restraint code = 2 in *RESTRAINTS). If a degree of freedom is specified more than once, then its displacement is the sum of that specified on separate lines.

Columns	Notes	Variable	Data
1(C)			"D", to indicate degrees of freedom.
4-5(I)			Ground displacement number.
10(I)			Direction code. (a) 1 : X translation. (b) 2 : Y translation. (c) 3 : Z translation. (d) 4 : X rotation. (e) 5 : Y rotation. (f) 6 : Z rotation.
11-20(R)			Scale factor for the displacements specified on this line. Default = 1.0.
21-30(I)		NF	First node.
31-40(I)		NL	Last node. Default = NF.
41-50(I)		ND	Node number difference for successive nodes. Default = all intermediate nodes between NF and NF.

*DISR : Resume Ground Displacement Analysis

The preceding analysis segment must have been *DISN or *DISR. The same ground displacement records, as for the preceding *DISN or *DISR analysis are used.

(a) Analysis Title.

Place title in columns 41-80 of *DISR line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Time at the end of segment will be time at beginning plus this increment.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 0 : continue with same KAUTO and same time step as preceding analysis segment. (b) 1 : change to constant time step. (c) 2 : change to variable time step.
21-30(R)		DT	Optional time step, as follows (KAUTO = 1 or 2 only) : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.

*FORN : New Dynamic Force Analysis

The structure must be in static state at the start of this analysis (the preceding analysis segment, if any, must have been *REST or *STAT or *GRAV).

This analysis is performed step-by-step through time. The time step may have a specified constant value, or may be varied automatically as the analysis proceeds. Within each time step an event-to-event strategy can be specified if desired.

(a) Analysis Title.

Place title in columns 41-80 of *FORN line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Segment starts at time zero and ends at time SEGTM.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 1 : constant time step. (b) 2 : variable time step.
21-30(R)		DT	Optional time step, as follows : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.

Continued

(c) Dynamic Force Records.

One line per applied dynamic force record (max. 36). Each record may be associated with one or more nodal degrees of freedom in (d). To represent a traveling wave, use the same record with different time delays. The applied records are numbered in the order of input.

Columns	Notes	Variable	Data
1(C)			"R", to indicate dynamic force record.
7-10(C)			Record name (see *FORCREC).
11-20(R)			Time scale factor. Default = 1.0.
21-30(R)			Force scale factor for the record. Default = 1.0.
31-40(R)		DTIME	Time delay.

Continued

(d) Degrees of Freedom.

These degrees of freedom must be free (restraint code = 0 in *RESTRAINTS) and non-slaved (see *SLAVING). If a degree of freedom is specified more than once, then its force will be the sum of that specified on separate lines.

Columns	Notes	Variable	Data
1(C)			"D", to indicate degrees of freedom.
4-5(I)			Dynamic force number.
10(I)			Direction code. (a) 1 : X translation. (b) 2 : Y translation. (c) 3 : Z translation. (d) 4 : X rotation. (e) 5 : Y rotation. (f) 6 : Z rotation.
11-20(R)			Scale factor for the forces specified on this line. Default = 1.0.
21-30(I)		NF	First node.
31-40(I)		NL	Last node. Default = NF.
41-50(I)		ND	Node number difference for successive nodes. Default = all intermediate nodes between NF and NF.

*FORR : Resume Dynamic Force Analysis

The preceding analysis segment must have been *FORN or *FORR. The same dynamic force records, as for the preceding *FORN or *FORR analysis are used.

(a) Analysis Title.

Place title in columns 41-80 of *FORR line.

(b) Control Information.

One line.

Columns	Notes	Variable	Data
1-10(R)		SEGTM	Time increment for this analysis segment. No default. Time at the end of segment will be time at beginning plus this increment.
11-15(I)		NSTEPS	Maximum number of time steps (analysis quits if exceeded). Default = 1000.
20(I)		KAUTO	Time step code. (a) 0 : continue with same KAUTO and same time step as preceding analysis segment. (b) 1 : change to constant time step. (c) 2 : change to variable time step.
21-30(R)		DT	Optional time step, as follows (KAUTO = 1 or 2 only) : (a) if KAUTO = 1 : constant time step for this analysis segment. (b) if KAUTO = 2 : initial time step. Default = value specified on DT line of *PARAMETERS.

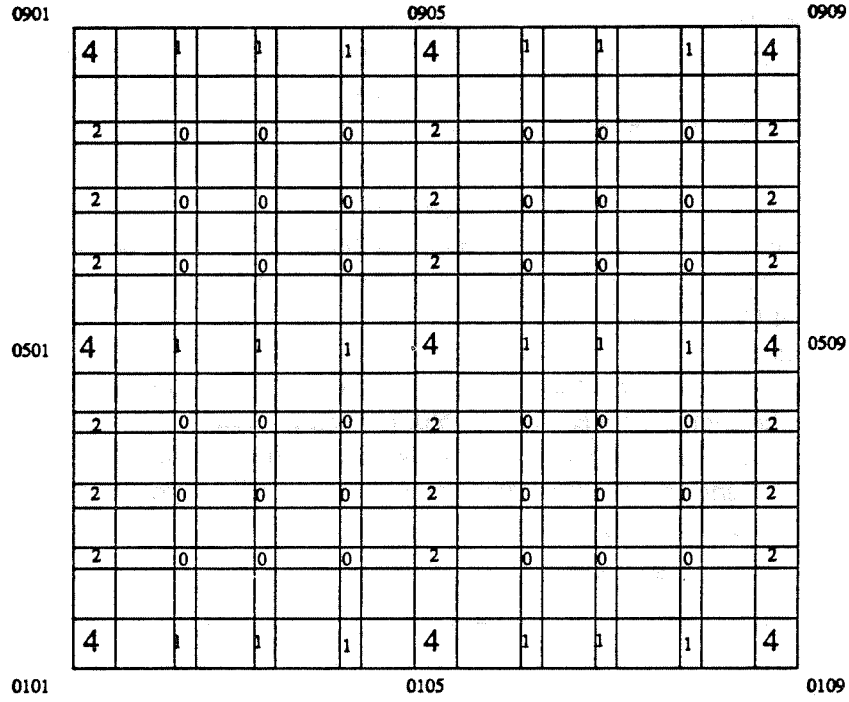
***STOP : Stop the Analysis Session**

This marks the end of the analysis session.

GENERAL NOTES

G1. Sequential generation is useful for specifying all nodes between NF and NL by defaulting on ND. In case defaulting on ND is not used, its capability can be duplicated and in fact superseded by the grid generation option. Typically, therefore this option will be first used to set a specification for all nodes, after which the specification for the deviant nodes will be reset by using the grid generation option.

G2. Grid generation is natural for nodes lying at the intersections of a physical grid. However, the program treats the grid as logical rather than physical and advantage can be taken of this in specification. In case the last specification prevails, it may be useful to first specify on finer grids, then on progressively coarser grids.



In the above example node types have to be specified. The nodes are of type 0, 1, 2 or 4. The nodes of type 0 are automatically specified by initial default. The following lines specify the node of types 1, 2 and 4.

NT	NB	NE1	ND1	NE2	ND2
1	0101	0109	1	0901	400
2	0101	0109	4	0901	100
4	0101	0109	4	0901	400

G3. A pattern or record is identified by its type (e.g. static element load) and its 4-character name. The name must be unique within its type, but may be re-used if desired for a different type.

*START/*RESTART : NOTES

S1. Each problem must be assigned a unique, 1-8 character name. Several permanent files are created for each problem, with file names of the form YYYYYYYY.XXX, where YYYYYYYY is the problem name, and XXX is a 3-character extension identifying the type of data in the file.

S2. A structure will typically be analyzed for several different loadings, carried out in a number of analysis sessions. In any analysis session, analyses can be performed for any number of analysis segments, each segment corresponding to a static load increment or the application of a dynamic load for a period of time. The analysis segments are numbered in sequence, with the unstressed state assigned a segment number of 0. The structure state at the end of any analysis segment can be saved on a permanent file, if desired.

In any analysis session, the initial state for each analysis segment except the first is always the state at the end of the preceding segment. The initial state for the first analysis segment can be any previously saved state (identified by its analysis segment number), not necessarily the latest state. This feature allows unlimited flexibility in choosing load sequences. In particular, static and dynamic segments can be inter-mixed. However, a static gravity segment must begin from the unstressed state (NSTAT = 0).

S3. The option KEXE = 1 is provided for data checking. The data is read, checked and echo printed, but analyses are not performed. The minimum required blank common length is computed, and also the blank common length for "in-core" execution. The option KEXE = 2 is to avoid inefficient execution if there is not enough memory for "in-core" execution. During execution of the program, data is stored for each element to monitor its nonlinear behavior. If the data for all elements can be stored in memory, then the program will execute most rapidly. If there is insufficient memory, the data is blocked, and the blocks must be recalled from disk for each step of the analysis. The additional I/O cost can be substantial, and execution time can be greatly increased. If KEXE = 2 and the program does not execute, it will be necessary to change the following statement in the MAIN.FOR file.

PARAMETER (NTSTP=N)

where N = number of integer words of memory to be allocated. On a virtual memory system, it will usually be most efficient to specify NTSTP sufficiently large to ensure "in-core" execution (i.e. virtual memory paging to disk will usually be more efficient than FORTRAN input-output to disk).

S4. The input lines are always echoed to the .ECH file. However, if a fatal error occurs, the program may crash and the current contents of the output buffer may not be written to the .ECH file. In such a case, it might be difficult to find which input line caused the crash. If the input echo code is set to 1, each input line is echoed to the monitor screen, as well as written to the .ECH file, as it is read. The last line read before the crash will thus be known.

Caution - this feature may not be useful in batch mode, as the "screen" output will be redirected to a file, which may also be buffered. It is recommended that this feature not be used in batch mode.

S5. This code is considered only for analysis sessions beginning from the unstressed state. For analysis sessions which begin from any other state, the code from the preceding session is automatically used (i.e. once turned on, P-delta effects can not be turned off, and vice versa). P-delta effects are considered only for element groups with KGEM greater than zero (see *ELEMENTGROUP).

S6. This code is considered only for analysis sessions beginning from the unstressed state. For analysis sessions which begin from any other state, the code from the preceding session is automatically used (i.e. once turned on, energy calculation can not be turned off, and vice versa). Allowing for energy calculations results in the setting up of a .SLO file, containing a log showing the energy balance and unbalanced loads.

*COMPOUND : NOTES

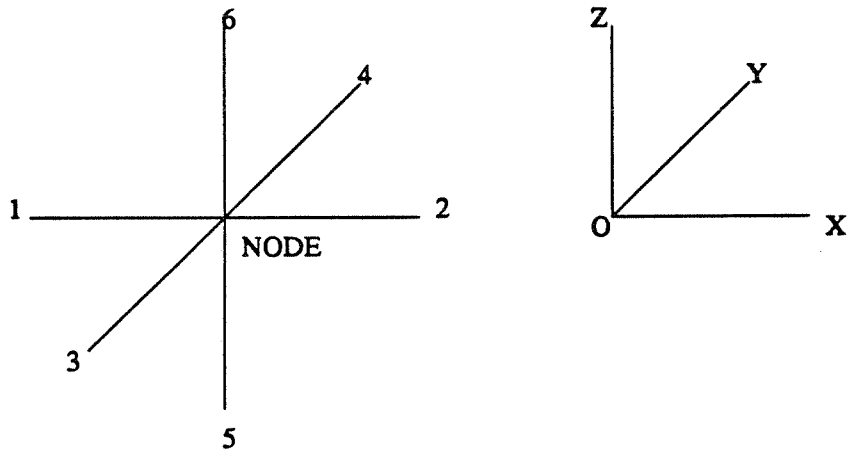
S1. A compound node consists of a main node and one or more subnodes. A compound node type defines the subnode locations relative to the main node, and also specifies information on the subnode degrees of freedom (dofs). The main node is identified for each compound node instance, and is not a part of the type definition.

Compound nodes will typically be used for connection elements (e.g. complex beam-to-column connections allowing for effects such as panel zone deformation, bond slip, etc.).

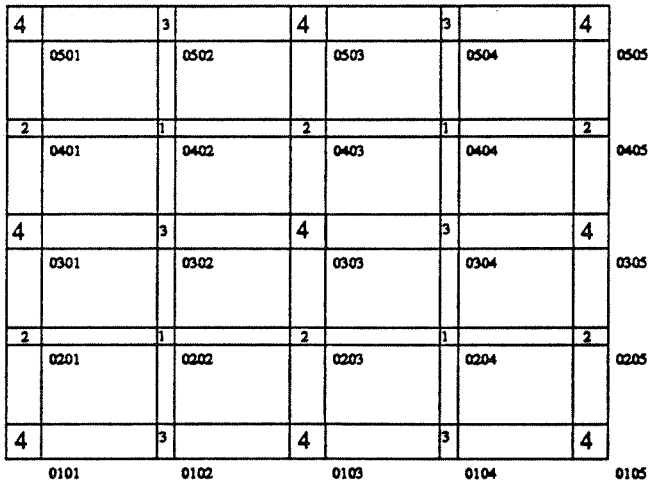
Provision is made for different types of subnode dofs in order to allow for different connection element models. Typically the default restraint code (=0) will be specified for subnode dofs. However, the connection element model might be based on displacements relative to the displacements of the main node in order to reduce numerical sensitivity when the connection is very stiff. A connection element might also be based on dofs which are not simple translations and rotations, for example, warping dofs for modelling torsion. Restraint codes equal to 2 or 3 account for these possibilities. The type of connection element to be used must be known when the compound node type is defined. If the dof types defined for a compound node do not agree with the dof types used by an element which connects to the node, an error will be indicated.

Typically in a building the compound node will be used to specify an entire joint (e.g. a beam-column joint) region. The joint will consist of a node which is located in the plane of the floor diaphragm, and a number of subnodes which are located at the faces of column and beam.

The analyst will typically follow a convention for numbering the subnodes around the node. Example :



The node would be referenced by a node number, which typically would indicate its location in the structure. Example :

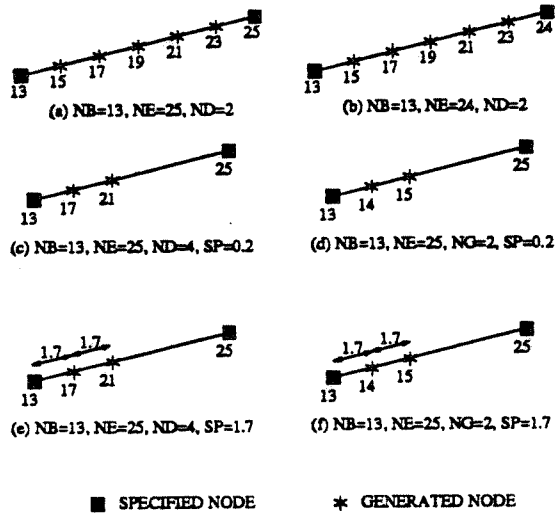


In the above example, the first two digits of the node number increase in the Y direction and the last two digits increase in the X direction. Thus the node number, can give an idea of the physical location of the node in the structure.

*NODECOORDS : NOTES

S1. For "control nodes" either absolute coordinates or relative coordinates (offsets) from a previously defined node can be input.

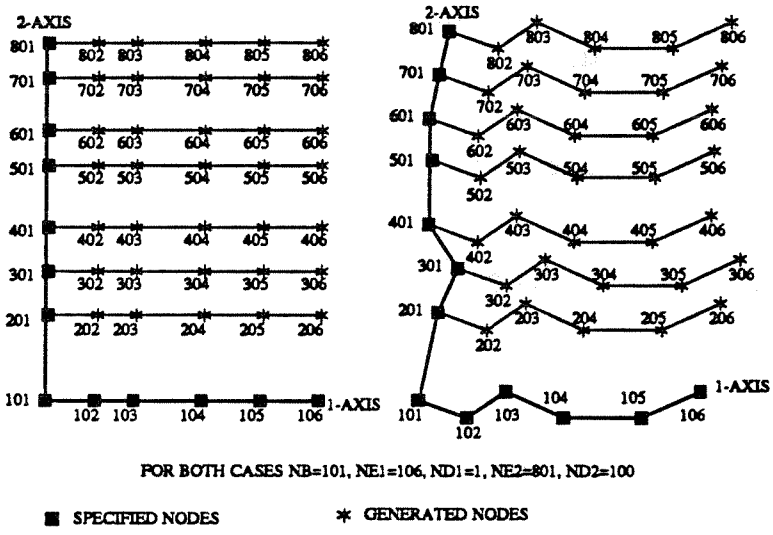
S2. Straight line generation is useful for generating equally spaced nodes along a straight line passing through NB and NE.



STRAIGHT LINE GENERATION EXAMPLES

S3. The frontal extrapolation allows nodes along a series of parallel grid lines to be generated from node data defining two adjacent sides of the grid. This option can be useful if the nodes follow a rectangular grid pattern. However, the method is not restricted to a rectangular grid. As the examples below show.

In the example below nodes 101,102,...106 define the grid 1-axis, and nodes 101,201,...801 define the grid 2-axis. In one case the nodes follow a rectangular grid pattern, in the other they do not.



FRONTAL EXTRAPOLATION EXAMPLES

The coordinates of a node at grid location (i,j) are calculated as :

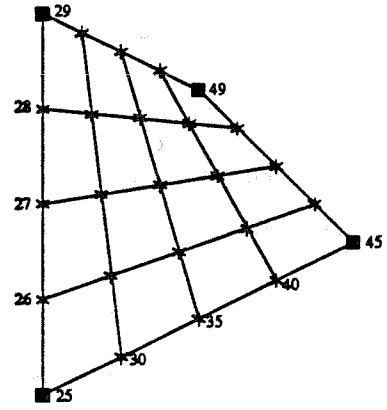
$$X(i,j) = X(i,0) + X(0,j) - X(0,0) \quad \text{for } i=1,2,..,(NE1-NB)/ND1$$
$$Y(i,j) = Y(i,0) + Y(0,j) - Y(0,0) \quad \text{and } j=1,2,..,(NE2-NB)/ND2$$
$$Z(i,j) = Z(i,0) + Z(0,j) - Z(0,0)$$

where,

X(0,0), Y(0,0) & Z(0,0) are coordinates at grid origin NB,
X(i,0), Y(i,0) & Z(i,0) are coordinates of node (NB + ixND1) on grid 1-axis, and
X(0,j), Y(0,j) & Z(0,j) are coordinates of node (NB + jxND2) on grid 2-axis.

The 1-axis and 2-axis will typically be horizontal and/or vertical. However, they may be along inclined directions if desired.

S4. Grid interpolation allows a series of nodes to be generated within a quadrilateral, given the nodes at the four corners. The generated nodes along the sides of the quadrilateral are equally spaced. The generated nodes inside the quadrilateral lie at the intersections of the grid lines joining corresponding nodes on opposite sides of the quadrilateral.



GRID INTERPOLATION EXAMPLE

*NODETYPES

S1. All nodes except compound nodes have a node type number equal to 0. Compound nodes have node type numbers between 1 and the number of compound node types. The compound node types are defined using the *COMPOUND separator and are numbered in input sequence.

*RESTRAINTS

S1. A restraint code = 1 means that the displacement magnitude is identically zero (the displacement is not a structure degree of freedom). A code = 2 means that the displacement is restrained by a stiff support spring (which is added automatically). The difference is significant only for dynamic ground displacement analyses (***DISN** or ***DISR** analysis). For these analyses, code = 1 means zero absolute displacement, whereas code = 2 means essentially zero displacement relative to the ground. Support points through which ground displacement will be introduced into the structure must have code = 2, not code = 1.

In a typical frame with imposed dynamic displacements, the restraint codes at all support points will = 2, and at all other points = 0. However, for a trussed frame, with elements which provide no rotational stiffness at the nodes, all rotational dofs will have restraint codes = 1, and at support points the translational dofs will have restraint codes = 2.

*SLAVING

S1. Slaved displacements of a node are obtained from the displacements of the master node, assuming a rigid link between the two. Master nodes can not be slaved (i.e. only "single level" slaving is permitted).

If a slaved dof has mass, this mass is automatically transferred to the master node, as follows.

(a) Translational X mass on slaved dof.

Add X mass to translational X mass of master node.

Add X mass times square of the Y offset to the rotational Z mass of the master node.

Add X mass times square of the Z offset to the rotational Y mass of the master node.

(b) Translational Y mass on slaved dof.

Add Y mass to translational Y mass of master node.

Add Y mass times square of the Z offset to the rotational X mass of the master node.

Add Y mass times square of the X offset to the rotational Z mass of the master node.

(c) Translational Z mass on slaved dof.

Add Z mass to translational Z mass of master node.

Add Z mass times square of the X offset to the rotational Y mass of the master node.

Add Z mass times square of the Y offset to the rotational X mass of the master node.

(d) Rotational X mass on slaved dof.

Add X mass to rotational X mass of master node.

(e) Rotational X mass on slaved dof.

Add Y mass to rotational Y mass of master node.

(f) Rotational Z mass on slaved dof.

Add Z mass to rotational Z mass of master node.

Note that the transfer of mass for translational dofs is not strictly correct, because it ignores off-diagonal contributions to the mass matrix of the master node. Hence, when a master node is chosen, it should be at (or at least close to) the mass centroid of the slaved nodes.

*MASSES

S1. The structure mass is assumed to be lumped at the nodes, with a diagonal mass matrix.

S2. If desired, a different mass proportional damping factor (α) can be specified for each mass. Typically, the same factor will be used for all masses, and this factor then needs to be specified in only the first line. If a node appears more than once in these lines, the αM value is the sum of the values for the separate lines. For slaved dofs, αM is transferred to the master node, in the same way that mass is transferred (see *SLAVING note).

*ELEMENTGROUP

S1. For input and output, the elements must be divided into groups. All elements in any group must be of the same type, and typically all elements of any type will be included in a single group. However, elements of the same type may be divided into separate groups if desired (for example, to separate the elements for output purposes, to allow different amounts of damping to be specified, etc.).

S2. Each element type in the element library is identified by a type number. For example, element type 1 is a truss element and element type 2 is a beam column element. See the separate element user guides for details. All elements in a group must be of same type. Element groups may be arranged in any convenient order. Within any group, the elements must be numbered in sequence, beginning with 1.

S3. An event-to-event solution strategy is automatically used for static analysis, and can be specified as an option for dynamic analysis. In this strategy, the structure stiffness is updated every time there is a significant change in stiffness for any element (i.e. an "event"). The option KEVE = 0 suppresses the event calculations for all elements in this group, for both static and dynamic analyses. In general this should be done only for elements which are specified to be very strong in order to force them to remain linear. Significant computer time can be saved, since the event factor calculation is omitted for these elements.

S4. P-delta effects will typically be negligible in girders, but may be important for columns. These effects are included in the analysis by adding geometric stiffnesses. The option KGEM = 1 allows the geometric stiffness to change progressively during static analysis but keeps it constant for dynamic analysis. This will often be accurate for building frames with P-delta effects in only the columns, since the translational geometric stiffness for a story depends only on the sum of the axial forces in all columns at the story, which is a constant equal to the gravity load if there is no vertical inertia and/or no vertical earthquake. The story translational geometric stiffness changes if there is vertical inertia, but the changes can probably be ignored. The story torsional geometric stiffness for a 3D building does depend on the individual column forces. However, this effect is probably small, and can probably be ignored.

If KGEM = 1, the geometric stiffnesses used for dynamic analysis are based on the element axial forces at the end of the last static analysis. If KGEM = 2, the geometric stiffnesses are reformed, but only for those elements which have stiffness change events. That is, if there are no stiffness change events in any time step, the geometric stiffness is not modified. If some elements have stiffness change events, the geometric stiffness are modified for those elements only. If it is necessary to modify the geometric stiffness at every time step, it will be necessary to use elements for which a change in axial force is assumed to be an event. Most of the elements do not make this assumption.

If desired, second order effects can be suppressed, even though KGEM is nonzero for some or all element groups (see KPDEL in *START/*RESTART).

S5. Each element has a constant viscous damping matrix equal to βK_0, where K_0 is the initial elastic stiffness matrix of the element (not including geometric stiffness). The value of β is same for all elements in any group, but can vary from group to group (e.g to allow different damping to be specified for different materials).

*SECTION : NOTES

S1. The procedure for defining a structure section is as follows:

(1) Draw a plane through the structure. Choose the three mutually perpendicular axes of the section plane. Axis 1 is perpendicular to the section plane. Axes 1, 2 & 3 must form a right handed system. Identify the elements which are cut by the plane and the locations of the cuts.

(2) Define the "center" of the section. The section moments will be calculated about this point.

(3) Calculate the distances from the center of the section to each element cut, along the 2 & 3 axes. All distances are measured from the center of the section to the element cut, positive in the positive directions of the section axes.

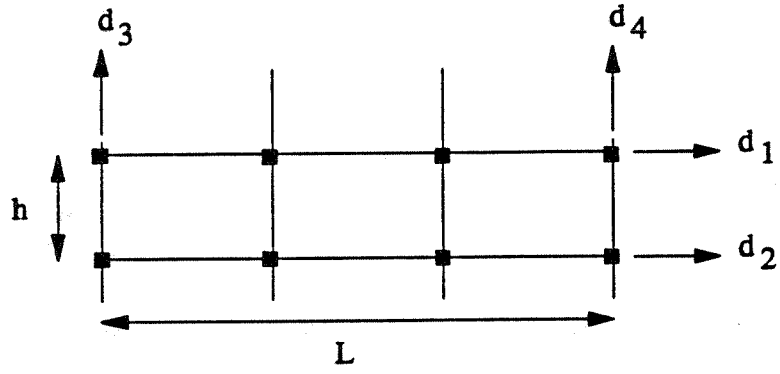
(4) At each element cut there will be, for any state of the structure, a set of 6 element forces. These consist of 3 forces and 3 moments in the global X, Y and Z coordinate system. The section forces and moment are obtained by (a) transforming these element forces to the section center, and (b) summing over all cut elements.

(5) The 6 forces at a cut in an element depend on the element end forces (or the nodal forces for a solid element) and the location of the cut in the element. Once the forces at the cut have been calculated, the program automatically transfers them to the section center and sums them to get section forces. However, the forces at the cut are not determined automatically. Instead, a force transformation which gives the 6 forces at the cut in terms of the forces at the element ends, must be input. The size of the transformation matrix for any element is 6*NDOF, where NDOF is the number of element degrees of freedom (e.g., for a beam-column element the size of the transformation is 6*12). Usually the transformation will be the same for several elements, and only a small number of transformations will need to be input. Also, since cuts will usually be made adjacent to nodes, the transformation matrices will usually be quite simple. See note 2 for more details.

S2. The transformation must transform nodal forces (X, Y & Z forces and X, Y & Z moments) acting on the element to forces (X,Y & Z forces and X,Y & Z moments) acting on the element section at the cut. Elements will typically be cut at nodes, in which case the transformation coefficients will be either unity or zero. Since there are two sides to each element cut, the signs of the transformation coefficients will depend on which side is assumed to be retained and which is removed. It is recommended that the part of the structure on the positive 1-axis side is assumed to be retained, and that the forces on the element cut be forces acting on the retained part of the element. The section forces are then forces acting on the retained part of the structure.

*GENDISP : NOTES

S1. A generalized displacement will typically define a relative displacement between two nodes, or a deformation based on the relative displacements of several nodes. For example, story drift in the X direction may be defined as the X displacement of a node on the floor above minus the X displacement of a node on the floor below. However, this definition of drift ignores the effect of column shortening, and may exaggerate the actual story deformation. Hence, story drift might alternatively be defined as an effective shear deformation of a story, as shown in the following figure :



Effective Deformation = $d_1 - d_2 - h/L(d_3 - d_4)$

The required deformation quantity is calculated as the sum of displacements of a number (up to a max. of 8) of degrees of freedom multiplied by corresponding participation factors. The displacements are specified, by specification of node number and direction.

***RESULTS : NOTES**

S1. This separator defines the results which are to be written to the .RXX post-processing file, and to the .OUT printout file, where XX is the analysis segment number. Post-processing of results written to the RXX file may be done using the DRAIN-POST program. The steps at which post-processing and printout occur can be different. These steps are defined in the *PARAMETERS section.

S2. For compound nodes, only the main node displacements are printed. However, the main node and subnode displacements are saved for post-processing.

***ELEMLOAD : NOTES**

S1. Element loads will typically account for gravity loads (dead loads or live loads) acting along element lengths. However, they may also account for thermal effects, or any other effects which originate within elements. Element loads can be applied only in "static gravity" analysis segments. These must begin with the unstressed state, and in the present version of the program nonlinear behavior is not permitted.

S2. Because the nature of the loading and its effects on an element depend on the element type, different input data is needed for each element type. Some element types do not have provision for element loads. Refer to the element user guides for details.

*ACCNREC : NOTES

S1. The input times are multiplied by this factor f_t. The effect is to increase the ground velocities by f_t, to increase the ground displacements by f_t^2, and to alter the frequency content of the earthquake. Time should not be scaled without carefully considering the influence on the ground motion.

S2. The input accelerations are multiplied by this factor. This factor will normally be used to convert the units of the acceleration values (e.g. from multiplies of "g" to in/sec^2). Further scaling can be specified when a time history analysis is actually performed.

*SPECTRUM : NOTES

S1. The input periods or frequencies are multiplied by this factor. This factor will normally be used to convert to units of Hz (cps) if frequencies are specified and to units of seconds if periods are specified.

S2. The input response values are multiplied by this factor. This factor will normally be used to convert the units of the response values. Further scaling can be specified when a response spectrum analysis is actually performed.

*NODALVEL : NOTES

S1. Initial velocity patterns can be used to test the energy absorbing capacity and/or impact response of a structure. Initial velocities corresponding to a known kinetic energy or a set of known impulses can be defined. With the velocities as initial conditions, a dynamic analysis can be performed. The analysis must begin from a static, but not necessarily unstressed state.

S2. Initial velocities can not be specified for slaved displacements.

*DISPREC : NOTES

S1. Ground displacement records are used for dynamic analyses with supports moving out of phase. There are no restrictions on the sequences in which ground displacements are applied, or on whether or not the structure remains linear. In-phase support motions can be considered as a special case.

S2. The input times are multiplied by this factor f_t. The effect is to reduce the ground velocities by f_t, to reduce the ground accelerations by f_t^2, and to alter the frequency content of the earthquake. Time should not be scaled without carefully considering the influence on the ground motion.

S3. The input displacements are multiplied by this factor. This factor will normally be used to convert to the units of the displacement values. Further scaling can be specified when a displacement time history analysis is actually performed.

*PARAMETERS : NOTES

S1. The structure state may be saved at the end of any analysis segment. The state is saved in a .SXX file, where S indicates a state file, and XX is the analysis segment number. Analysis segments are numbered in sequence, regardless of whether or not they are saved.

For nonlinear static and dynamic analyses, the state may also be saved at step intervals and/or or time intervals within an analysis segment. If this is done, a new file is not created each time the state is saved. Instead a new file is created the first time that the state is saved, but at each subsequent time that file is rewound and over-written. Hence, if the analysis segment runs to completion, the file will contain the state at the end of segment. If, however, execution is terminated for some reason before the segment is completed, the file will contain the most recently saved state.

S2. A .RXX file can be created for post-processing of the results by the DRAIN-POST program, where R indicates a post processing result file, and XX is the analysis segment number. This file contains results for selected nodes, elements, generalized displacements and structure sections (see *RESULTS). For dynamic analysis, results may be saved at IPPSVD time step intervals or TPPSVD time intervals, whereas for static analysis the results can be saved at IPPSVS load step intervals or at every event (IPPSVS = -1). Hence, the file can be very large. It may be necessary to run (and post process) a large problem in several short analysis segments.

S3. Results for selected nodal displacements, elements, generalized displacements, and structure sections (see *RESULTS) may be printed to the .OUT file for each analysis segment. For dynamic analysis, results may be printed at IPOUTD time step intervals or TPOUTD time intervals, whereas for static analysis the results can be printed at IPOUTS load step intervals or at every event (IPOUTS = -1). To print results at the end of analysis segments only, input large values for IPOUTS, IPOUTD, and TPOUTD.

S4. A .EXX file can be created for post processing of the results by the DRAIN-POST program, where E indicates a post processing envelope file, and XX is the analysis segment number. Envelope values include maximum positive and negative values for all node displacements, selected response quantities for all elements (varies with element type), all generalized displacements and all structure section forces. The time or load step at which the maximum occurred is also output for each value. For dynamic analysis, results may be saved at IENVSD time step intervals or TENVSD time intervals, whereas for static analysis the results can be saved at IENVSS load step intervals. To save envelopes at the end of the analysis segment only, input large values for IENVSS, IENVSD and TENVPD.

S5. Envelopes for all nodal displacements, elements, generalized displacements, and structure sections may be written to the .OUT file for each analysis segment. For dynamic analysis, envelopes may be printed at IENVPD time step intervals or TENVPD time intervals, whereas for static analysis the envelopes can be printed at IENVPS load step intervals. To print envelopes at the end of the analysis segment only, input very large values for IENVPS, IENVPD and TENVPD.

S6. In a step-by-step dynamic analysis, energy balance and/or equilibrium may not be satisfied at the end of a time step. Energy balance can be satisfied by modifying the velocities at the end of a time step, and equilibrium can be satisfied by modifying the accelerations. These corrections will usually improve the accuracy. For more details, see the theory report. Events are not necessarily calculated for all element groups (see *ELEMENTGROUP).

S7. In a solution scheme with event calculations, a large number of events in a single time step may indicate a flip-flop situation. If MAXEV is exceeded, the analysis may be terminated or it may be continued, omitting calculation of any further events. Termination is usually recommended, since a large number of events in a time step usually indicates too long a time step or other problems.

S8. Four dynamic solution schemes are available, namely constant (KAUTO=0) time step with (KEVNT=1) and without (KEVNT=0) event calculation, and variable (KAUTO=1) time step with (KEVNT=1) and without (KEVNT=0) event calculation. KAUTO is specified for each dynamic analysis segment. In the variable time step scheme, the time step is controlled by upper and lower tolerances on the "static force" and "inertia force" errors (see DRAIN-2DX theory report). If either TOLHIS or TOLHII (see *PARAMETERS (h)) is exceeded in a step, the time step is decreased by multiplying the current time step by DTRED, and the step is repeated. The time step is not reduced below a specified minimum value (DTMIN). If both errors are less than their lower tolerances for IICONT consecutive steps, the time step is multiplied by DTINC (for the next time step). The time step is not increased above a specified maximum value (DTMAX). DTMAX must be chosen to ensure that ground accelerations or other dynamic loads are accurately represented. See the theory report for guidance in selecting tolerances.

*STAT : NOTES

S1. Each load pattern is multiplied by its scale factor, and added to the load increment. The load increment defines a unit loading, i.e. a loading corresponding to a load factor equal to unity.

The load increment is initialized to zero for the first *STAT analysis, or if the preceding analysis was not a *STAT analysis, or if the first contributing load pattern name is not blank. If the preceding analysis was a *STAT analysis, and the first pattern name is blank, then the load increment is initialized to the unit loading for the preceding *STAT analysis multiplied by the specified pattern scale factor.

S2. If there is no displacement control, load factor increments of this magnitude will be applied until the load factor increment for the analysis segment is reached.

If displacement control is also specified, then the sign of the step load factor is determined by displacement control.

S3. Displacement control is useful if the structure becomes very flexible, and is essential if the structure is to be loaded into the post buckling range. If the structure becomes unstable, a positive load increment may cause a negative displacement, leading to a flip-flop condition (successive unloading-reloading). Displacemnent control can be used to force the structure to continue displacing in the positive direction, even though the load decreases. If there is no displacement control, the analysis quits if the structure becomes unstable.

S4. Even if a displacement control is specified, the solution can flip-flop. This will be indicated by successive changes in the sign of the load increment required to make the controlled displacement increase progressively. This flip-flop limit can avoid waste of computing time.

S5. If both load control and displacement control are specified, then the number of steps for analysis completion cannot generally be calculated in advance, as the step load factor magnitude is minimum of that required by load control and by displacement control. Also the load factor sign may be sometimes positive and sometimes negative in order that the controlled displacement always increases. Thus a very large number of steps may be taken before the segment increments for the load factor or displacement are reached for analysis completion. This maximum number of steps limit, can avoid waste of computing time.