Computational Mechanics Lab Manual

Corporate

Schaeffer Enterprises
harry.schaeffer@yahoo.com

Disclaimer

The concepts, methods and examples presented in this text are for illustrative and educational purpose only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. Schaeffer Enterprises assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Notices

All restrictions that apply to this software product apply to any third party products, documentation, or code which are contained in this software product. No copies may be made or distributed in any form without the prior written permission of its owners or suppliers.

This software product contains a DES Implementation, which conforms to MIT's libdes. All rights, title and interest in the DES Implementation code and related documentation are and shall remain in Eric Young.

Table of Contents

1 EXECUTIVE CONTROL SECTION

Installation

I assume you have received a zipped file contains the NASTRAN-CORE installation. I also assume that you know how to extract the contents to a directory of your choosing. Let's denote the installation directory by the variable, CML_INSTALLDIR.

Set Environment Variables (Windows)

In the Control Panel, open System -> Advanced ->Environment Variables. In the top paned that is labeled User Variables, enter the following Name, Value pairs:

Name	Value
CML_INSTALLDIR	<name of directory>
CML_RCFILE	<name of directory>/RC/nast.rc
CML_RFDIR	<name of directory>/RF

Then, in the bottom panel add <name of directory>/BIN to the PATH

Test Installation

Open a Terminal window and enter the command

```
core.exe <CML_INSTALLDIR>\examples\beam_sol_1.dat
```

If all went well the program will write several comments to the terminal window that summarizes the start of the program. The line TMG_OUT tells us where to look for the output file. In this case the .out file is located in the <CML_INSTALLDIR>/examples.

Example - Using DMAP to Solve the Test Problem

The test problem included an ALTER that modified the DMAP script for SOL 101 to write the KLL and PL data blocks in Harwell Boeing format using the OUTPUTBH Module. The Alter is:

```
ALTER 210

OUTPUTBH KLL,PL,,, // C,N,41/C,N,42 $

ENDALTER
```

Looking at the DMAP manual one notes the Data Block names in the input section and two constant Parameters having values of 41 and 42 that are the fortran units associated with KLL and PL, respectively where KLL is the stiffness matrix and PL is the load matrix. The test problem also included the following ASSIGN statements in the Executive Control Section:

```
ASSIGN OUTPUTBH='C:/NAST_CORE/BIN/BEAM_SOL_1.KLL',UNIT=41 ,
```

```
 form=formatted
ASSIGN OUTPUTBH='C:/NAST_CORE/BIN/BEAM_SOL_1.PL',UNIT=42 ,
 form=formatted
```

These statements define the FULL path name, the Fortran UNIT and its FORM of the OUTPUTBH file. It must be notes that the blank space followed by the comma at the end of the first line define a continuation of the ASSIGN statement. So both of these files are formatted (The default us UNFORMATTED) as you can verify by using a text editor to open them.

The input file for the solution using the test file OUTPUTBH is /examples/beam_solve.dat:

```
id solve,inputtbh
ASSIGN inputtbh='C:/NAST_CORE/BIN/BEAM_SOL_1.KLL',UNIT=41 ,
 form=formatted
ASSIGN inputtbh='C:/NAST_CORE/BIN/BEAM_SOL_1.PL',UNIT=42 ,
 form=formatted
app dmap
diag 4,8,14
begin $
inputtbh /kll,pl,,, / c,n,41/c,n,42 $
solve    kll,pl / ulv $
matprn kll,pl,,, // $
matprn ulv,,,,//c,n,0/c,n,1 $
end $
cend
title=Read Harwell Boeing KLL and PL matrices, solve and print results
begin bulk
enddata
```

Notice:

1. The ASSIGN now define the files to be associated with the INPUTTBH Module.
2. The APPROACH is DMAP (NOT DISPLACEMANT)
3. The DMAP script starts with begin $and ends with end $
4. INPUTTBH reads in the KLL and PL data blocks
5. SOLVE performs sparse Gauss Reduction to solve for the Displacement data block, ULV
6. MATPRN prints ULV to the .out file

Now, run this file and compare the results with those of the test run. Perhaps you can convince your self that they are the same.

Running a Problem

The RUN Command

core *input_filename   keyword=value*

Where: *input_filename* is the file containing the input data.The input file can have any name. If the file does not exist the code will search for the same file name with ".dat", ".inp" or ".bdf" extensions. If there are any blank spaces in the *input_filename*, then the entire filename must be placed within a set of double quotation marks.

Initial Operations

Prior to performing the solution steps defined in the DMAP script file that is associated with the SOL statement in the Executive Control Section the program performs a number of operations including:

1. Processing program arguments defined by the Keywords and values
2. Automatically creating filenames for output files
3. Process User-defined Fortran files
4. Setting the memory configuration

Automatic Filenames

These files include

Table 1-1 Automatically Generated Files

Filename	File Number	Description
Input file	5	The input file full path name for the problem that is defined by the first argument
Output file	6	The output file called <input>.out (called F06 in MSC Nastran)
Log file	3	The log file called <input>.log
OP2 file	12	File created by the OUTPUT2 module in all SOLs, called <input>.op2. The SOL DMAP automatically assign this file to file number 12
OP4 file	14	File created by the OUTPUT4 module, called <input>.op4. Not automatically included in SOL DMAP. Can be included as an ALTER but the P2 Parameter of the module must be 14.
NPTP	8	The New problem tape

User-Defined Files

Generally these files are associated with using INPUTT(i) and OUTPUT(i) Modules that are defined using either an ALTER to an existing SOL or in a user-defined DMAP script

These files include

Table 1-2 User Files

Keyword	Value	File Number
FTN11	<user-defined>	11
FTN12	<user-defined>	12
FTN13	<user-defined>	13
FTN14	<user-defined>	14
FTN15	<user-defined>	15
FTN16	<user-defined>	16
FTN17	<user-defined>	17
FTN18	<user-defined>	18
FTN19	<user-defined>	19
FTN20	<user-defined>	20
FTN21	<user-defined>	21
FTN22	<user-defined>	22
FTN23	<user-defined>	23
...		
FTN100	<user-defined>	100

Keywords

1. The name associated with *keyword* is case insensitive. Keywords can be typed as they appear in **Table 1-3**.
2. If there is an space in the value of a keyword, then it should be placed between a pair of double quotation marks
3. Variables between < > in the list below can be ignored
4. Default values for all arguments on the command line may be set in the runtime configuration file.
5. Defaults for the keywords can also be specified by the file defined by the CML_RCFILE environmental variable..

Table 1-3 Keywords

#	Keyword	Description	Default
1	JID	JID=<Path>&Filename Job ID an optional keyword to set the inputfile.	No default
2	EXE	EXE=Path&Filename Path to an alternative executable file.	[Installed Directory]default executable
3	MEMORY	MEMORY=XXX <K, M, G, K, W, MW, GW, KB, MB or GB> XXX words or memory value will be assigned to the run. K=Killo,M=Mega,G=Giga, W=Words and B=Bytes. One Word= 4 Bytes. Maximum allowable Memory is the minimum of 500 MW (2.0GB) and the amount allowed by the OS' resources. The minimum required memory is 1MW.	20000000 words or 80000KB or 80MB or 0.080GB

Table 1-3 Keywords

#	Keyword	Description	Default
4	NPTP	NPTP=Path&<FileName> Set the path to NPTP file. If the path was defined, the NPTP file will be the inputfile's name with NPTP extension and will be created in the path defined by user. If path&filename was defined, the NPTP file will be the defined filename, created in the path defined by user.	[outputpath]outputfile.nptp
5	OLD	OLD=Y<YES> or N<NO> If OLD=YES is set, then old copies of output files will be saved. It will rename the existing output files in a sequential form by adding a ".i" at the end of the output files. The largest i indicate the latest output prior to the current run.	YES
6	OP2	OP2=Path&<FileName> Set the path to OP2 file. If the path was defined, the OP2 file will be the inputfile's name with OP2 extension and will be created in the path defined by user. If path&filename was defined, the OP2 file will be the defined filename, created in the path defined by user.	[outputpath]outputfile.op2
7	OP4	OP4=Path&<FileName> Set the path to OP4 file. If the path was defined, the OP4 file will be the inputfile's name with OP4 extension and will be created in the path defined by user. If path&filename was defined, the OP4 file will be the defined filename, created in the path defined by user.	[outputpath]outputfile.op4
8	OPTP	OPTP=<Path>&FileName Set the path to OPTP file which is used to restart from a previous check point. The NPTP from the previous run must be renamed with the OPTP extension.If this keyword is used, this file must exist.	[No Default Value]
9	OUTPUT	OUPUT=<Path>&<FileName> Direct the outputs to a defined value. If user defines a path, then all the outputs will be created in that path using the input filename. If user defined just filename then base of all output files will be set to defined filename and located to the path user defined.	Path where the user launch nastran and input file name as base.
10	PS	PS=Y<YES> or N<NO>. Allow the user to choose postscript format when a plot request has been used in input file.	YES
11	PLT1	PLT1=Y<YES> or N<NO> Allow the user to choose PLT1 format if a plot request has been used in input file.	NO

Table 1-3 Keywords

#	Keyword	Description	Default
12	REAL	REAL=nn An Integer number, indicating the percentage of the memory to be set for open core.Acceptable range: 20 <= 95.	80
13	RFDIR or DELIVERY	RFDIR=Path or DELIVERY=Path These two keywords have same functionality. With these keywords user can address alternative set of Rigid Formats (solution sequences).	[Installation Path]\rf
14	SCRATCH	SCRATCH=Y<YES> or N<NO>. SCRATCH=Y will erase the database files (".dic" & ".nptp") at end of the run.	NO
15	SDIR	SDIR=Path Will set the path to the scratch directory. Scratch Directory will be erased after completion of a run.	[Default OS temp directory]
16	SILENCE	SILENCE=Y<YES> or N<NO>. SILENCE=YES will disable all interaction between user and program. Errors and warnings will be printed in log files.	NO
17	SOF or SOF1 or SOF2	SOF=Path&Filename Assigns SOF files, for substructuring. SOF and SOF1 are equivalent.	No Default Value
18	SYSTEM	System Cell Setting: SYSnnn=mmm or SYSTEM(nnn)=mmm (system cell nnn has value of mmm) or system_cell_name=mmm (system cell with the name "system_cell_name" has a value of mmm)	
19	BATCH (Unix only)	BATCH=Y<YES> or N<NO> If BATCH=YES the program runs in the background	

Input File

The input file is a text file that contains lines of tagged data statements where the format of the statement is associated with the tag. The file has the following ordered sections (depending on the particular job requirements):

- The NASTRAN statement (optional)
- The Executive Control Section (required)
- The Substructure Control Section (required only in substructure analyses)
- The Case Control Section (required)
- The Bulk Data Deck (required)
- The INPUT Module Data data statement(s) (required *only* if the INPUT module is used).

The NASTRAN Statement is used to change the default values for certain operational parameters, such as the buffer size and the machine configuration. It is optional, but, if present, it must be the first data statement of the NASTRAN input file. It is described in detail in the **NASTRAN Statement** (p. 10).

The Executive Control Section (ECS) begins with an optional ID statement and is terminated by the required CEND statement. ECS identifies the type of solution to be performed and declares the general conditions under which the run is to be executed, such as, maximum time allowed, type of system diagnostics desired, restart conditions, and whether or not the run is to be checkpointed. If the run is to be executed using one of the included solution sequences (which are traditionally called Rigid Formats (RF)), the rigid format is specified along with any alterations to the rigid format as appropriate. If Direct Matrix Abstraction (DMAP) is used, the complete DMAP sequence must appear in the Executive Control Section. The executive control cards and examples of their use are described in **EXECUTIVE CONTROL SECTION** (p. 1). If the ID statement is not included as the first statement, the ID is given a default value.

If Automated Multi-stage Substructuring (AMSS) is included then the Substructure Control Section begins after the ECC with a SUBSTRUCTURE statement and terminates with a ENDSUBS statement. It defines the general attributes of the Automated Multi-stage Substructuring capability and establishes the control of the Substructure Operating File (SOF). The associated commands are described in Chapter 3.

When AMSS is not included, the Case Control Section (CCS) begins after CEND statement and is terminated by the BEGIN BULK entry. It defines the subcase structure for the problem, makes selections from the Bulk Data Section, and makes output requests for printing, punching and plotting. A general discussion of the functions of Case Control and a detailed description of the associated data statements used in this section are given in Chapter 3

The Bulk Data Section begins after BEGIN BULK and is terminated by ENDDATA. It contains all of the details of the structural model and the parameters for the solution. A detailed description of all of the bulk data entries is given in **Bulk Data Entry Descriptions** (p. 222). The BEGIN BULK and ENDDATA entries must be present even if no bulk data entries are specified.

Generally, only one structural model can be defined in the Bulk Data Section. However multiple sets of some of the bulk data entries such as those associated with loading conditions, constraints, direct input matrices, transfer functions, and thermal fields may exist. Only specific set members selected in Case Control will be used in any particular solution.

Comments, which begin with a dollar sign ($) may be inserted in any of the parts of the input file.

Usage of Secondary Input Files Via the READFILE/INCLUDE Capability

Data files can be imported into the nastran file by using either the INCLUDE or READFILE directives which are alternative names for the same capability. In the following description the INCLUDE directive is used. That name can be replaced by READFILE.

Description of the Capability

The format of the INCLUDE command is as follows:

```
INCLUDE <name>
```

where <name> is the full path name of an external text file. If the file is named grids.dat and is in the C:\user\widget directory, then the include command would be:

```
INCLUDE c:\user\widget\grids.dat
```

When an INCLUDE is encountered in the primary input file, the input file processor reads all subsequent input from the specified secondary file until an end-of-file condition or an ENDDATA statement is encountered on that file, whichever occurs first. If an end-of-file condition is encountered on the secondary file before an ENDDATA is detected, the program resumes reading of the input data from the primary input file and the process continues. If an ENDDATA is encountered on the secondary file before an end-of-file condition is detected, input file processing is terminated.

The features of the INCLUDE command are:

- The format of statement is free-field. The only restrictions are that there should be at least one space between the tag, INCLUDE, and the <name> of the secondary file and that the length of the statement must be 80 characters or less, and there are no included blanks in <name>.
- Nested INCLUDE commands are allowed. That is, INCLUDE commands are permitted in both the NASTRAN primary and secondary input files.
- The NOPRINT option turns off the echo of data statements from the INCLUDE file.
- INCLUDE commands may be used anywhere in the Executive Control, Substructure Control, Case Control and Bulk Data Sections. (The NASTRAN statement can also be specified in a secondary file.)

Examples of READFILE Capability Usage

The following example illustrates several ways in which the READFILE capability can be used.

```
SOL 1
...
BEGIN BULK
INCLUDE c:\b707\FUSELAGE.DT
READFILE c:\b707\WINGS.DT
INCLUDE,NOPRINT,c:\b707\TAIL.DT
ENDDATA
```

NASTRAN Statement

Many of the important operational parameters used in NASTRAN-CORE, such as the size of the read/write buffer and the machine configuration, are saved internally. These and other operational parameters are initially assigned values by the program. However, the program does provide a means by which the default values initially set for some of these operational parameters can be redefined at execution time using the NASTRAN statement, or alternatively on the command line of the launch script described in the previous section.

The NASTRAN statement is optional, but, if used, it must be the first statement of the input file. It is free-field card (similar to the cards in the Executive and Case Control Decks). If the same keywords are set by both the NASTRAN statement and the launch script, the value specified by the launch script is used.

The format of the statement is:

```
NASTRAN keyword1 = value,  keyword2 = value, ...
```

The list of applicable and acceptable keywords is as follows:

BUFFSIZE — Changes the first word in /SYSTEM/. This word defines the number of words in the Input/Output buffer associated with each file. The default value is BUFFSIZE=1024

The desired value at a particular installation may be different from the default value. In any event, related runs such as restarts must use the same BUFFSIZE for all parts of the run.

BULKDATA — Changes the 77th word in /SYSTEM/. This parameter specifies whether NASTRAN is to run normally (BULKDATA = 0, the default) or if NASTRAN is to terminate after the Preface (or Link 1) operations (BULKDATA not equal to 0).

BULKDATA = -3 (a special option) indicates the NASTRAN 15 GINO timing constants are to be calculated, printed, and the run is terminated.

FILES — Establishes the specified NASTRAN files as executive files. The files that may be specified are POOL, NPTP, OPTP, NUMF, PLT1, PLT2, INPT, INP1, INP2,...INP9. Multiple file names must be specified by enclosing them in parentheses, such as FILES = (PLT1, NPTP). The FILES parameter, if used, must be the last keyword.

HICORE — Changes the 31st word in /SYSTEM/. Defines the total amount of memory allocated to the process, in words. (See also REAL)

MAXFILES — Changes the 29th word in /SYSTEM/. This word defines the maximum number of files to be placed in COMMON /XFIAT/ by subroutine GNFIAT. The default value is 35.

MAXOPEN — Changes the 30th word in /SYSTEM/. This word defines the maximum number of files that may be open at any one time in the program. The default value is 16.

NLINES — Changes the 9th word in /SYSTEM/. This word defines the number of data lines per printed page. The smallest acceptable value is 10. The default value is 55. Alternatively, the number of data lines per printed page can also be defined by means of the LINE statement in the Case Control Section.

PLOTOPT — Defines the action to be taken by NASTRAN in the case where plots are requested and error(s) exists in the Bulk Data Section. The default is zero (PLOTOPT = 0) if the PLT2 file is not assigned in a NASTRAN job and one (PLOTOPT = 1) if the PLT2 file has been assigned. the plot options (0 through 5) are listed below:

PLOTOPT	Bulk Data	Plot Commands	NASTRAN Action
0	No error	no error	Executes, no plots.
	No error	error	Stops after data check.
	Error	Error or no error	Stops data check.
1	No error	No error	Executes and plots.
	No error	Error	Stops data check.
	Error	Error or no error	Stops a data check.
2	Error or no error	No Error	Stops after undeformed structural plots.
	Error or no error	Error	Stops data check.
3	Error or no error	Error or no error	Attempts to plot; then stops.
4	No error	No error	Executes and plots.
	No error	Error	Attempts to plot then stops.
	Error	No error	Stops after undeformed structural. plots.
	Error	Error	Stops data check.
5	No error	No error	Executes and plots.
	No error	Error	Executes but no plots.
	Error	No error	Stops after undeformed structural plots.
	Error	Error	Stops after data check.

SYSTEM(J) — Changes the Jth word ($1 \le J \le 100$) in /SYSTEM/. This is the general form of defining any word in /SYSTEM/. For some values of J, SYSTEM(J) has equivalent keywords. For example, SYSTEM(1) and BUFFSIZE are equivalent and SYSTEM(9) and NLINES are equivalent.

REAL — An Integer number, indicating the percentage of the memory to be set for open core. It can be as low as 20 up to 95.

Examples

Following are some examples of the use of the NASTRAN statement.

`NASTRAN BUFFSIZE = 900`	changes the 1st word of /SYSTEM/
NASTRAN NLINES = 40	changes the 9th word of /SYSTEM/.
NASTRAN FILES = (PLT1, NPTP)	establishes the PLT1 and NPTP files as executive files.
NASTRAN HICORE = 50000000, REAL=80	set the total memory allocated to the process to 50 Million words (200MB), sets open core to 40 Million words and sets the DISK to 10 Million words

Documented System Cells

Many system cells which control program action can be set by the user. These include:r

Table 1-4 Documented System Cells
(* indicates system cells which should not be changed)

System Cell Number	Keyword	Default	Description
1	BUFFSIZE	8192	GINO buffer size
2	F06	6	FORTRAN unit for out file
4*	MPC	none	MPC set from current subcase
5*	SPC	none	SPC set from current subcase
7	F04	3	FORTRAN unit number for log file
8*	LOAD	none	pointer to first record in case control for current subcase
9	NLINES	none	Number of lines per page in the f06 file
11*	NCPAGES	none	Current page count
12*	NCLINES	none	Current number of lines on current page
13*	TLINES	none	Total number of lines printed
14	MAXLINES	2147483647	Maximum number of lines allowed in f06 file
15-17*	DATE	none	Date when job started
18*	TIMEW	none	Time (seconds) past midnite 1/1/2000 when job started
20	PLOTOPT	0	Nonzero value indicates request for structural plots
21*	APPRCH	2	Approach for current run 1 = force 2 = disp 3 = DMAP negative = restart
28	PRINTSYS	0	a nonzero value will print the user-modifiable system cell descriptions in the f06 file
31	HICORE	20000000	Length of open core in words
32*	TIMEZ	none	Time (seconds) since midnite when job started
34*	MAJOR	none	Major release number
35*	MINOR	none	Minor release number
36*	MICRO	none	Micro release number (patch level)
42-44*	SYSDAT	none	Three BCD words containing month, day, and year of system generation date
45*	TAPFLG	32	Indicates whether files are on tape or disc - Bits are turned on counting from the right representing the files in the xxfiat. The default setting is used to indicate that the plt1 file is to be written using ASCII, rather than binary - this should not be changed by the user.
55*	IPREC	none	Precision flag 1 = single precision 2 = double precision
58*			Selects old (non-sparse) MPYAD
70	STST	-2	Exponent for singularity test in EMG

Table 1-4 Documented System Cells
(* indicates system cells which should not be changed)

System Cell Number	Keyword	Default	Description
74*	ITIME	none	Problem start time
75*	CTIME	none	print flag for DMAP sequence
79	DIAGA	0	Alternate way to set DIAG 1 through 32
80	DIAGB	0	Alternate way to set DIAG 33through 64
81	DIAGC	0	Alternate way to set DIAG 65 through 96
88	K88	0	reserved for user
89	K89	0	reserved for user
90	K90	0	reserved for user
101	SPARSE	1	0 disables sparse matrix operations, 1 enables sparse matrix operations
103	WRT2SCRN	0	nonzero value sends CONMSG output to screen
104	MSGLVL	0	Controls the message level in the decomposition and eigenvalue routines. The default is system(104) = 0. Increasing the value increases the amount of diagnostic printout. Normally users would not override the default. Allowable values: 1-4
105	MAXRAT	1	controls MAXRATIO calculation and reporting in decomposition 0 = no action 1 = print messages for dof where ratio of diagonal to diagonal factor is >= MAXRATIO and add them to the partitioning vector generated 2 = print message for (1) and any terms with negative terms on factor diagonal matrix - add them ot the partitioning vector generated 3 = (2) plus messages for any 0.0 diagonal terms encountered
106	MORDER	0	Reordering method selection 0 = both (pick the better of the two) 1= MMD 2= METIS
107	IREVERT	0	revert to non-sparse method if sparse solver has insufficient memory 0 = issue FATAL error and terminate if insufficient memory for sparse solver 1 = issue WARNING message and revert to conventional solver is insufficient memory for sparse solver
110	CHKBULK	3 (0 if substruc-turing or restarts are used)	Bulk data checking 0,1 = no checks performed 2 = perform checks and issue WARNING if required data is not present 3 = perform checks and issue FATAL if required data is not present
111	NRANK	0	Rank order for sparse solver
112	NBLOCK	0	Right hand block size for sparse solver
113	NRATIO	0	Row/column ratio for math kernel blocking logic

Table 1-4 Documented System Cells
(* indicates system cells which should not be changed)

System Cell Number	Keyword	Default	Description
114	NOSEID	0	0 = terminate run if SEID specified on GRID or GRDSET entry 1 = ignore SEID on GRID and GRDSET entries
115	SPMPYAD	1	0 = regular MPYAD 1 = sparse MPYAD
116	SPUDCMP	1	0 = non-sparse sparse unsymmetric decomp 1 = sparse umsymmetric decomp
117	DPUDCMP	-3	Exponent of diagonal pivot in sparse unsymmetric decomp
118	OPUDCMP	-1	Exponent of off-diagonal pivot in sparse unsymmetric decomp
119	SVDSPC	1	0 = old AUTOSPC 1 = new (Singular Value Decomposition) AUTOSPC
120	NODETJ	0	Controls action if if a negative jacobian is found 0 = issue FATAL error and terminate run 1 = ignore and continue (not recommended)
121	ZEROTOL	0	Filter control for SDR1 - a non-zero value will filter the displacements (and velocities and accelerations in transient response) and GPKE by changing any terms with a magnitude less than $1.x10^{SYSTEM(121)}$ to 0.0 (or 0.0,0.0 for complex numbers)
122	COREDUMP	0	A non-zero value for this system cell will result in a buffer dump to the f06 file for some I/O related error messages.
123	ESEFILT	-10	Filter for ESE/EKE/EDE calculated energy values - any value less than $10^{SYSTEM(123)}$ will be set to 0.0
125	SOLVER	0	Choose default solver for eigenvalue solution (See executive control SOLVER command) 0 = Lanczos (or method requested on EIGR/EIGRL) 1= SMS 2 = Rose's Method
126	BADMATL	0	A non-zero value will force the program to continue although one or more materials provided result in 0.0 for G11 and G22 when processing QUAD elements.
127	RUNTIME	1	Controls generation of .rtm (Run Time Monitor) file. A negative value will disable this feature.
129	FLUSH	-1	Interval (DMAP lines) between buffer flusihng for output files. A negative value indicates that the program will not flush the buffers.
131	BUSHCIDA	0	If BUSHCIDA has a non-zero value, the element coordinate system for BUSH elements will be taken as the displacement coordinate system at GRID A
132	GAPSP	100000001	Initial SPOINT to be generated for initial opening for GAP entries

Executive Control Statements

Executive Control statements are free field. **The name of the operation (for example, CHKPNT) is separated from the operand by one or more blank characters**. The fields in the operand are separated by commas, and may be up to 8 integers or alphanumeric as indicated in the control statement descriptions. The first character of an alphanumeric field must be alphabetic, followed by up to 7 additional alphanumeric characters. Blank characters may be placed adjacent to separating commas if desired.

The following Executive Control statements are required under certain circumstances:

ALTER	Specifies the insertion and or deletion of following DMAP statements
APP	Specifies an approach in a solution sequence
ASSIGN	Assigns a physical file for use in the current run
BEGIN$	Specifies the beginning of user-provided DMAP statements.
CEND	Terminates Executive Control.
CHKPNT	Requests the execution be checkpointed.
DIAG	Requests diagnostic output or modifies operational parameters.
END$	Specifies the end of user-provided DMAP statements.
ENDALTER	Specifies the end of user-provided changes to a solution sequence.
GEOMCHECK	Controls element geometry checking.
ID	Optional, if present is the first statement of the Executive Control Section.
RESTART	Specifies the beginning of a restart dictionary.
SOL	Specifies the solution number of a Rigid Format.
SOLVER	Select eigenvalue solver
TIME	defines the maximum execution time in minutes.
$	defines a non-executable comment.

Executive Control Section Examples

1. Cold start, no checkpoint, rigid format, diagnostic output.

```
APP          DISPLACEMENT
SOL          2,0
TIME         5
DIAG         1,2
CEND
```

2. Cold start, checkpoint, rigid format.

```
CHKPNT       YES
APP          DISPLACEMENT
SOL          1,3
TIME         15
CEND
```

3. Restart, no checkpoint, rigid format. The restart dictionary indicated by the double line bracket is automatically punched on previous run in which the user selected the CHKPNT option.

```
RESTART PERNAST, SPACECFT, 09/11/01, 18936,
1, XVPS, FLAGS=0, REEL=l, FILE=6
2, REENTER AT DMAP SEQUENCE NUMBER 7
3, GPL, FLAGS=0, REEL=1, FILE=7
$ END OF CHECKPOINT DICTIONARY
APP          DISPLACEMENT
SOL          3,3
TIME         10
CEND
```

4. Cold start, no checkpoint, DMAP. User-written DMAP program is indicated by double line brackets.

```
APP          DMAP
BEGIN $
...
INCLUDE C:\USER\DEMAP\NEWSOL.DAT
END $
TIME       8      CEND
```

5. Restart, checkpoint, altered rigid format, diagnostic output.

```
RESTART BEAM, FREE, 09/11/01, 77400,
1, XVPS, FLAGS=0, REEL=1, FILE=6
2, REENTER AT DMAP SEQUENCE NUMBER 7
3, GPL, FLAGS=0, REEL=1, FILE=7
$ END OF CHECKPOINT DICTIONARY
CHKPNT       YES
DIAG         2,4
APP          DISPLACEMENT
SOL          3,3
TIME         15
ALTER        20 $
MATPRN       KGGX,,,,// $
TABPT        GPST,,,,// $
ENDALTER
CEND
```

Executive Control Format

The format is free-field. In presenting general formats for each statement embodying all options, the following conventions are used:

1. Upper-case letters and parentheses must be includes as shown.
2. Lower-case letters indicate that a substitution must be made.
3. Double brackets indicate that a choice of contents is mandatory. Brackets [ ] contain an option that may be omitted or included.
4. First listed options or values are the default values.
5. Text lines must be 72 or less characters in length
6. Continuation statements can be used to represent data statements requiring more than 72 characters

ALTER - *DMAP Sequence Alteration Request*

Description

Requests Direct Matrix Abstraction Program (DMAP) sequence of a Rigid Format to be changed by additions, deletions, or substitutions.

Format and Examples

```
ALTER K1 [,K2]$
ALTER 22 $
ALTER 5,5 $
ALTER 38,45 $
ALTER 25,19 $
```

Option	Meaning
K1	DMAP statement number after which DMAP instructions following the ALTER to be inserted. Integer > 0.
K1 and K2	DMAP statement numbers identifying a single DMAP statement or a range of DMAP statements to be deleted and replaced by any DMAP statements following the ALTER. Integer > 0. See remark 5.

Remarks

1. See the descriptions of the INSERT and DELETE statements for alternate ways of specifying DMAP sequence alteration requests.
2. The DMAP statements referenced on ALTER, INSERT and DELETE statements (either explicitly or implicitly, when a range is specified) must be referenced in ascending order of their occurrence in the rigid format DMAP.
3. Use DIAG 14 to obtain a listing for the DMAP program to be altered.
4. If both K1 and K2 are specified and K1 is not equal to K2, a range of DMAP statements is implied and either of them can be less than the other. If K1 = K2, a single DMAP statement is implied.

ASSIGN - *Assigns a Physical File for use in the current run*

Description

Assigns file names for use in the current run - allows you to create, open, or delete files.

Format and Examples

ASSIGN Logical_Key='filename', UNIT=u,

$$\left[\left[STATUS = \begin{bmatrix} NEW \\ OLD \\ UNKNOWN \end{bmatrix} , FORM = \begin{bmatrix} FORMATTED \\ UNFORMATTED \end{bmatrix} \right., \textbf{TEMP, DELETE} \right]$$

```
ASSIGN OUTPUT2='myrun.op2', UNIT=21, DELETE
ASSIGN INPUTT4='mydata.ip4', UNIT=31, FORMATTED
ASSIGN NPTP=myrun.nptp
ASSIGN NEWDICT=myrun.dict
ASSIGN SOF1=test.sof
```

Remarks

1. Default values for STATUS and FORM depend on the Logical_Key

Logical_Key	Default STATUS	Default FORM	Default Location
CDICT	NEW	FORMATTED	OUT
INPUTTBH	OLD	UNFORMATTED	IN
INPUTT2	OLD	UNFORMATTED	IN
INPUTT4	OLD	UNFORMATTED	IN
NDICT	NEW	FORMATTED	IN
NEWDICT	NEW	FORMATTED	OUT
NPTP	NEW	N.A.	OUT
OLDDICT	OLD	FORMATTED	IN
OPTP	OLD	N.A.	IN
OUTPUTBH	NEW	UNFORMATTED	OUT
OUTPUT2	NEW	UNFORMATTED	OUT
OUTPUT4	NEW	UNFORMATTED	OUT
OP2	NEW	UNFORMATTED	OUT
OP4	NEW	UNFORMATTED	OUT
SOF, SOF1-SOF10	UNKNOWN	UNFORMATTED	OUT
USERFILE	UNKNOWN	none	IN

2. If no directory path is specified in 'filename', the default location in the above table is used. IN = directory containing the input file. OUT = the directory defined by "OUT=" on the command line (defaults to IN).

3. CDICT, NDICT, and NEWDICT are used to assign the checkpoint dictionary file if CHKPNT YES is specified. These options will ignore UNIT, STATUS, FORM, TEMP, and DELETE on the ASSIGN statement.
4. OLDDICT is used to assign the checkpoint dictionary from a previous run to a restart run. This Logical_Key results in the ASSIGN statement working identically to INCLUDE, NOPRINT. If used, it must be after the assignment for the OPTP has occured. When using this Logical_Key, the ASSIGN statement will ignore the UNIT, FORM, STATUS, TEMP, and DELETE options
5. OPTP is used to assign the “Old Problem TaPe” from a previous run to be used in a restart run. When using this Logical_Key, the ASSIGN statement will ignore the UNIT, FORM, STATUS, TEMP, and DELETE options.
6. NPTP is used to assign the “New Problem TaPe” to a run. When using this Logical_Key, the ASSIGN statement will ignore the UNIT, FORM, STATUS, TEMP, and DELETE options.
7. SOF, SOF1, SOF2, SOF3, SOF4, SOF5, SOF6, SOF7, SOF8, SOF9, and SOF10 are used to assign the SOF files to the run. SOF and SOF1 are identical, both assign SOF1. When using these Logical_Keys, the ASSIGN statement will ignore the UNIT, FORM, STATUS, TEMP, and DELETE options.
8. The logical ASSIGNstatement is allotted 4096 characters but each line is limited to 80 characters. Line are continued by ending the line with a blank followed by a comma as noted in the “Installation” section in this Chapter.
9. File unit numbers in NASTRAN-CORE - the following table describes the use of the unit numbers in the program. File units 11-100 are available for the user. You should avoid using any of the other unit numbers.

Table 1-5

File unit numbers and their usage

(* implies these units should not be used on an ASSIGN statement)

Unit Number	Use
1*	Punch file
2*	rc files, sysinfo.out, ifp.tab, nasinfo
3*	log file
4*	Dictionary file
5*	Input file
6*	out file
7*	Old problem tape (OPTP)
8*	New problem tape (NPTP)
9*	xpdt
10*	Plot file
11-100	available to the user
60-89	used during ifp, but are available for use on ASSIGN statement after ifp (no conflict will occur if you wish to use these numbers)
104*-198*	Solver scratch files
199*	conmsg file
200*-299*	Data set manager scratch files

BEGIN - *DMAP Sequence Initiation*

Description

Defines the beginning of a Direct Matrix Abstraction Program (DMAP) sequence.

Format and Examples

```
BEGIN $
BEGIN OPTIONAL NAME OF DMAP SEQUENCE $
```

Remarks

1. Must be the first statement of a DMAP sequence. The statement is included at the beginning of the DMAP sequence defining a Rigid Format.This statement, like all DMAP statements, is terminated with the $ character delimiter.
2. This statement is a non-executable instruction for the DMAP compiler.
3. The DMAP User’s Manual includes specific instructions related to DMAP usage.

CEND - *Executive Control Deck Terminator*

Description

Defines the end of the Executive Control Section.

Format and Examples

```
CEND
```

Remarks

1. Must be last statement in the Executive Control Section.

CHKDATA - *Request to check if required bulk data entries are present*

Description

Enables checking to verify that required bulk data entries are present at the start of the run

Format and Examples

```
CHKDATA /{YES, NO}/(FATAL, WARNING
CHKDATA YES
```

Remarks

1. This command is optional, but may be used to enable or disable bulk data checking. If checking is enabled, the program will check to see that the required bulk data is present for the solutions you request in case control.
2. If FATAL is chosen, the program will issue a FATAL message for each component of the bulk data which is missing, then terminate the run. If WARNING is chosen, the program will indicate that required bulk data is missing, but will continue the run.
3. The default is CHKDATA YES, FATAL, unless you are running substructures or using restarts, in which case, the default becomes CHKDATA NO

CHKPNT - *checkpoint file request*

Description

Requests data blocks be written to a checkpoint file for a later restart.

Format and Examples

```
CHKPNT /{YES, NO}/
CHKPNT YES
```

Remarks

1. This command is optional but when it is used, the checkpoint file (NPTP) must be made available using operating system control statements and the NASTRAN statement.
2. The restart dictionary is automatically written to the NPTP for use in a later restart execution.
3. AMSS provides an alternative but is an Advanced method

DELETE - *DMAP SEQUENCE ALTERATION REQUEST*

Description

Requests the Direct Matrix Abstraction Program (DMAP) sequence of a rigid format to be changed by deletions or substitutions.

Format and Examples

DELETE $s\text{-}mod_1$[,$s\text{-}mod_2$] $

where $s\text{-}mod_i$ is taken from the general format: $n\text{-}mod_i[(r_i)][,n_i]$

```
DELETE SSG1 $
DELETE EMA(2) $
DELETE READ,1 $
DELETE SDR2(2),-1 $
DELETE SSG3,REPT $
DELETE GP2,GP3,-1 $
DELETE SMA3,1,TA1,-1 $
DELETE REPT,2,REPT,3 $
```

Option	Meaning
$s\text{-}mod_i$	Nominal module (Alphanumeric value, no default). See Remark 5.
r_i	Occurrence flag (Integer > 0, default = 1). If $r_i > 0$, the r_i ‘th occurrence of the nominal module in the rigid format DMAP sequence (counting from the beginning of the DMAP sequence) defines the reference module. See Remark 6.
n_i	Offset flag (Integer, default = 0). The DMAP module that is offset from the reference module by n_i DMAP statements in the rigid format DMAP sequence defines the specified module. See Remark 7.
$s\text{-}mod_i$ only	Specified module defined as per the above scheme that is to be deleted and replaced by any DMAP instructions that may follow the DELETE statement.
$s\text{-}mod_i$ and range of specified modules	Defined as per the above scheme: 1 specmod that are to be deleted and replaced by any DMAP instructions 2 that may follow the DELETE specification.

Remarks

1. See the description of the **ALTER** statement for an alternate way of specifying DMAP sequence deletions and substitutions.
2. The DMAP statements referenced on ALTER, INSERT and DELETE cards (either explicitly or implicitly, when a range is specified) must be referenced in ascending order of their occurrence in the rigid format DMAP.
3. The nominal module nommod must be a valid name of a DMAP module in the rigid format DMAP sequence.
4. The default value of 1 for the occurrence flag r_i implies that the reference module is the first occurrence of the nominal module in the rigid format DMAP sequence.
5. The value of the offset flag n_i may be positive, negative or 0. A positive value means that the specified module follows the reference module by n_i DMAP statements in the rigid format DMAP sequence. A negative value indicates that the specified module precedes the reference module by n_i DMAP statements in the DMAP sequence. A value of 0 (the default) implies that the reference module is the specified module.

6. If both $s\text{-}mod_i$ and $s\text{-}mod_2$ are specified, it implies a range of DMAP statements and either of them can precede the other in the rigid format DMAP sequence.

DIAG - *Diagnostic Output and Operation Request*

Description

Requests additional information to be printed out or requests executive operations to be performed.

Format and Examples

DIAG $n[,-m]$

Option	Meaning
n	Is an integer number that defines an action as specified by the following table.

DIAG Number	Action
1	Dump memory when fatal message is generated.
2	Print File Allocation Table (FIAT) following each call to the File Allocator.
3	Print status of the Data Pool Dictionary (DPD) following each call to the Data Pool Housekeeper.
4	Print the Operation Sequence Control Array (OSCAR). See Remarks 3 and 7.
5	Not used
6	Not used
7	1. Print eigenvalue extraction diagnostics for real and complex determinant methods.
	2. Print statistics for Lanczos method
8	Print matrix and table data block trailers as they are generated.
9	Suppress echo of checkpoint dictionary. See Remark 7.
10	Use alternate nonlinear loading in TRD. Replace $\{N_{n+1}\}$ by 1/3 $\{N_{n+1} + N_n + N_{n-1}\}$.
11	Prints the number of passes in non sparse FBS
12	Controls diagnostics for complex eigenvalue analysis
13	Print length of open memory
14	Print the DMAP sequence that is compiled (NASTRAN SOURCE PROGRAM COMPILATION). See Remarks 3, 4, 5, and 8
15	Trace GINO OPEN/CLOSE operations.
16	Trace real inverse power eigenvalue extraction operations or eigensolution diagnostics for FEER tridiagonalization.
17	Print the compiled DMAP on the system PUNCH file
18	Not used
19	1. Print data for MPYAD method selection
	2. Print debug in TRNSP
20	Reserved for future use.
21	Print a table of all degrees of freedom in which the displacement sets to which it belongs are identified. See Remarks 6 and 7.
22	Print the displacement degrees of freedom that belong to user-specified displacement sets. See Remarks 6 and 7.
23	Print the DMAP ALTERs generated during Automated Multi-stage Substructuring. See Remark 7
24	Print the DMAP ALTERs generated during Automated Multi-stage Substructuring on the system PUNCH file. See Remark 6.
25	Print a cross reference listing of the DMAP program that is compiled. See Remarks 3, 4, and 6.
26	Revert plot FIND default to APR 1984 version.
27	Print the Input File Processor (IFP) table.
28	Reserved for future use
29	Reserved for future use

DIAG Number	Action
30	Reserved for future use
31	Print the module properties list (MPL) data.
32	Print a list of degrees of freedom. For each degree of freedom, the displacements sets to which it belongs are identified
33	Print the contents of various displacement sets. For each set, a list of degrees of freedom belonging to that set is given
34	Skip property ID, material ID, and coordinate ID cross-reference checking in the Preface.
35	Print machine hardware timing constants. (See NASTRAN BULKDATA = -3 option.)
36	Print internal and SIL (Scalar Index List) numbers for grid and scalar points vs. their external numbers. See Remark 6.
37	Suppress eigenvalue lower roots message (for real inverse power and FEER methods only).
38	Print element processing information during element matrix generation phase in EMG, DSMG1 and for Rigid and constraint elements
39	Print trace of eigenvalues for the PK method in flutter analysis.
40	Turn on diagnostic when layer composite material is used in PCOMP or PCOMPi entries. Includes listing the Bulk Data entries generated.
41	1. Writes matrix form in TRNSP
	2. Selects MPY routine
	3. Used in FEERX
42	Controls the output of USER POTENTIALLY FATAL messages: the default is that the messages is not printed; if DIAG 42 is specified, the messages is printed in the f04 file.
43	Skips speed improvements in MPYAD
44	Used in SDCOMP to revert to old decomp
45	1. Prints statistics in smcph1
	2. Prints warning in smchp2
46	Used to control calculation of consistent mass in obsolete elements
47	1. Prints statistics for sparse FBSI
	2. Used in HDPLOT
48	Print NASTRAN release news and the DIAG table. See Remark 6
49	Not used
50	Print a list of elements from the GPTA1BD block data file. This DIAG is for development use only. Note that the elements listed may or may not be supported.
51	Change error messages 2049 (undefined GRID on ASET/OMIT) and 2051 (undefined GRID in SPC set) from FATAL to WARNING

Remarks

1. One or more DIAG numbers may be chosen from the above table.
2. Multiple options may be selected by using multiple integers separated by commas or by using multiple DIAG statements.
3. See the description of the XDMAP statement in the DMAP User's Manual for alternate means of controlling the DMAP compiler options.
4. DIAG 14 is automatically turned on when DIAG 25 is requested.
5. The DMAP compiler default is set to LIST for restart runs and for runs using the DMAP approach (APP DMAP) or the substructure capability (APP DISP,SUBS). The default is also set to LIST when the REF option on the XDMAP statement is specified. The default is set to NOLIST for all other cases. There is, therefore, no need to use the DIAG 14 option in the former cases where LIST is the default; instead, the NOLIST option on the XDMAP statement can be used in these cases to suppress the automatic listing of the compiled DMAP program.

6. Use of any one or more of DIAGs 17, 21, 22, 24, 25, 28, 30, 36 and 48, in conjunction with DIAG 20, will result in job termination.
7. DIAG 21 and 22 are not recommended, as they result in the printout of the set definition tables before AUTOSPC processing is done. It is preferable to use PARAM,USETPRT.
8. If a DMAP alter is used, which results in warning or information messages, the program will stop execution unless DIAG 14 is set.

END - *DMAP Sequence Terminator*

Description

Defines the end of a Direct Matrix Abstraction Program (DMAP) sequence.

Format and Examples

```
END$
```

Remarks

1. Required at the end of a DMAP sequence. It must be the last statement. The statement is included at the end of the DMAP sequence defining a Rigid Format. when using the DMAP approach, it must be the last statement of a user-supplied DMAP sequence.
2. This statement, like all DMAP statements, is terminated with the $ character delimiter.
3. For specific instructions related to DMAP usage, **see Section 5.2**.
4. The END $ statement cannot be altered into a Rigid Format at intermediate steps. To schedule an early termination, use either the EXIT $ statement or the JUMP, FINIS $ statement.

ENDALTER - *Rigid Format DMAP Alter Terminator*

Description

Defines the end of a user supplied alter to a Rigid Format Direct Matrix Abstraction Program (DMAP) sequence.

Format and Examples

```
ENDALTER
```

Remarks

1. Required when an alter to a Rigid Format DMAP sequence is supplied if CEND is not the next statement.
2. If multiple alters are included, the ENDALTER terminates the alters.

GEOMCHK Control for element geometry checking

Description

Determines whether geometry checking is performed when processing the elements and controls the associated output.

Format and Examples

GEOMCHK, [MSGLIMIT = n],[MSGTYPE = $\left\{\frac{WARNING,}{FATAL}\right\}$], [SUMMARY], [NONE]

```
GEOMCHK, MSGTYPE=FATAL
GEOMCHK, SUMMARY
GEOMCHK, MSGLIMIT=0, SUMMARY
```

Remarks

1. This statement can be used in Executive Control Section only.
2. The default is GEOMCHK, NONE.
3. MSGLIMIT is the maximum number of messages which will be printed for each element type.
4. The GEOMCHK Bulk Data entry is used to control the acceptable tolerances when performing geometry checks on the elements.

ID - *Run identification for restart purposes*

Description

Specifies a comment at the start of the ECS.

Format and Examples

```
ID A1,A2
ID A1234567,B7654321
```

Option	Meaning
A1	Any alphanumeric field of eight or less characters, the first character of which must be an alphabetic character.
A2	Any alphanumeric field of eight or less characters, the first character of which must be an alphabetic character.

Remarks

1. The ID statement is optional.
2. If used in a restart run, it must be identical to the ID in the initial run, as it is used for verification that the correct OPTP file is attached.

INCLUDE - *Directive to Read Input statements*

Description

Defines a file that contains the input cards.

Format and Examples

```
INCLUDE, { PRINT,    } [ = ]  'filename'
         { (NOPRINT) }
```

(PRINT, is underlined as the default.)

```
INCLUDE 'c:\user\specialalter\myalter.alt'
INCLUDE NOPRINT 'c:\user\specialalter\myalter.alt'
INCLUDE, NOPRINT 'c:\user\specialalter\myalter.alt'
INCLUDE, NOPRINT, 'c:\user\specialalter\myalter.alt'
INCLUDE (NOPRINT) 'c:\user\specialalter\myalter.alt'
INCLUDE = 'c:\user\specialalter\myalter.alt'
INCLUDE NOPRINT  = 'c:\user\specialalter\myalter.alt'
INCLUDE, NOPRINT  = 'c:\user\specialalter\myalter.alt'
INCLUDE (NOPRINT) = 'c:\user\specialalter\myalter.alt'
```

Remarks

1. This statement can be used in Executive, Case Control, and Bulk Data Sections.
2. Input statements are saved in the file named filename.
3. Comma, equal sign, and parentheses are not allowed in filename.
4. NOPRINT allows reading in the input statements, such as the DMAP alters or restart dictionary, without printing them. The default PRINT.
5. Since this statement can also be used in the Case Control Section, an equal sign is also allowed.
6. Nested `INCLUDE` is allowed.
7. The punctuation in the pathed file name is system dependent.
8. READFILE is an alternative name for INCLUDE
9. If a relative path is provided, it will be relative the the directory containing the input file.
10. I If no path is specified, the directory will be the one containing the input file.

INSERT - *DMAP SEQUENCE ALTERATION REQUEST*

Description

Requests the Direct Matrix Abstraction Program (DMAP) sequence of a rigid format to be changed by additions.

Format and Examples

```
INSERT specmod $
```

where specmod has the following general form at: $n\text{-}mod[(r)][,n]$

```
INSERT GP4 $
INSERT EMA(2) $
INSERT READ,1 $
INSERT SDR2(2),-1 $
```

Option	Meaning
nommod	Nominal module (Alphanumeric value, no default). See Remark 5.
r	Occurrence flag (Integer > 0, default = 1). If r>0 the r'th occurrence of the nominal module in the rigid format DMAP sequence (counting from the beginning of the DMAP sequence) defines the reference module. See Remark 6.
n	Offset flag (Integer, default = 0). The DMAP module that is offset from the reference module by n DMAP statements in the rigid format DMAP sequence defines the specified module. See Remark 7.
specmod	Specified module defined as per the above scheme after which DMAP statements following the INSERT card are to be inserted.

Remarks

1. See the description of the ALTER specification for an alternate way of specifying DMAP sequence additions.
2. DMAP statements referenced on ALTER, INSERT and DELETE statements (either explicitly or implicitly, when a range is specified) must be referenced in ascending order of their occurrence in the rigid format DMAP.
3. The nominal module nommod must be a valid name of a DMAP module in the rigid format DMAP sequence.
4. The default value of 1 for the occurrence flag r implies that the reference module is the first occurrence of the nominal module in the rigid format DMAP sequence.
5. The value of the offset flag n may be positive, negative or 0. A positive value means that the specified module follows the reference module by n DMAP statements in the rigid format DMAP sequence. A negative value indicates that the specified module precedes the reference module by n DMAP statements in the DMAP sequence. A value of 0 (the default) implies that the reference module is the specified module.

READFILE - *Directive to Read Input statements*

Description

Defines a file that contains the input data.

Format and Examples

$$\text{READFILE, } \left\{ \begin{array}{c} \underline{\text{PRINT,}} \\ \langle\text{NOPRINT}\rangle \end{array} \right\} [\ =\ ] \quad \text{`filename'}$$

```
READFILE 'c:\user\specialalter\myalter.alt'
READFILE  NOPRINT 'c:\user\specialalter\myalter.alt'
READFILE, NOPRINT 'c:\user\specialalter\myalter.alt'
READFILE, NOPRINT, 'c:\user\specialalter\myalter.alt'
READFILE (NOPRINT) 'c:\user\specialalter\myalter.alt'
READFILE  = 'c:\user\specialalter\myalter.alt'
READFILE  NOPRINT  = 'c:\user\specialalter\myalter.alt'
READFILE, NOPRINT  = 'c:\user\specialalter\myalter.alt'
READFILE (NOPRINT) = 'c:\user\specialalter\myalter.alt'
```

Remarks

1. This statement can be used in Executive, Case Control, and Bulk Data Sections.
2. Input statements are saved in the file named filename.
3. Comma, equal sign, and parentheses are not allowed in filename.
4. NOPRINT allows reading in the input statements, such as the DMAP alters or restart dictionary, without printing them. The default PRINT.
5. Since this statement can also be used in the Case Control Section, an equal sign is also allowed.
6. Nested READFILE is allowed.
7. The punctuation in the pathed file name is system dependent.
8. INCLUDE is an alternative name for READFILE.

RESTART - *Restart Dictionary Initiator*

Description

Defines the beginning of a restart dictionary when reading data blocks from the previously checkpointed file.

Format and Examples

```
RESTART
RESTART ,
```

Remarks

1. The complete restart dictionary consists of this statement followed by one statement for each file checkpointed. The restart dictionary is automatically written to a file having the same filename as the input file with the extension, NPTP, when operating in the checkpoint mode. All subsequent statements are continuations of this logical statement. The entire dictionary file is required for a restart.
2. A restart is performed on a previous solution which generated both the RESTART directory and the NPTP. In addition to including the RESTART directory, the extension NPTP from the previous solution must be changed to OPTP for the restarted run. If the restart is also checkpointed it will generate an new NPTP. Obviously, some care must be exercised in managing files using this technology.
3. Each continuation begins with a sequence number. There are two types of continuations which are required and one that is not.

   Basic continuation statement:

   ```
   NO,DATABLOCK,FLAG=Y,REEL=Z,FILE=W
   ```

   where:

NO	Specifies the sequence number of the statement. The entire dictionary must be in sequence by this number.
DATABLOCK	Specifies the name of the data block referenced by this statement.
FLAG=Y	Defines the status of the data block where Y = 0 is the normal case and Y = 4 implies this data block is equivalenced to another data block. In this case (FLAG=4) the file number points to a previous data block which is the ‘actual’ copy of the data.
REEL=Z	An obsolete relic from the days of multi-reel files. REEL = 1, referring to the NPTP.
FILE=W	Specifies the internal file number of the data block on the OPTP. A zero value indicates the data block is purged. For example: `1,GPL,FLAGS=0,REEL=1,FILE=7` means that data block GPL is file 7 on the NPTP. `2,KGG,FLAGS=4,REEL=1,FILE=20` means that KGG is equivalenced to the data block which is file 20. (Note that FLAGS=4 cards usually occur in at least pairs as the equivalenced operation is at least binary). `3,USETD,FLAGS=0,REEL=1,FILE=0` implies USETD is purged.

   Re-entry point:

   ```
   NO,REENTER AT DMAP SEQUENCE NUMBER N
   ```

where:

NO	NO is the sequence number of the statement.
N	Specifies the sequence number associated with the DMAP instruction at which an unmodified restart will resume execution. There may be (generally, there are) several reentry statements in a restart dictionary, but only the last such statement is operative.

End of dictionary statement:

```
$ END OF CHECKPOINT DICTIONARY
```

is a comment that is written as a convenience to signal the end of the dictionary. It will not be written for terminations associated with non-NASTRAN failures.

4. The previously checkpointed file must be made available via operating system control statements.
5. A restart statement of the form

```
RESTART
```

can be used to read and process the OPTP associated with any previously checkpointed run. This file will have been called the NPTP in the previous run. The user will need to rename that file to a file having the OPTP extension.

6. A restart using the checkpointed file and dictionary created on a previous release of NASTRAN may not always be successful. First, the BUFFSIZE used on the later release may be different from that used on the earlier release. Second, any changes that might have been made to the rigid formats may effectively destroy the validity of the restart dictionary.

SOL - *Solution Number Selection*

Description

Selects the solution number which defines the Rigid Format.

Format and Examples

$$\text{SOL} \begin{Bmatrix} K1 \\ A \end{Bmatrix} \left[, \begin{matrix} K2 \\ \underline{0} \end{matrix} \right]$$

```
SOL 5
SOL 1,6
SOL 1,6,7,8,9
```

Option	Meaning
K1	Solution number of Rigid Format (see Remarks below and Volume II).
K2	Subset numbers for solution K1, default value = 0.
A	Name of solution sequence (see Remarks below).

Remarks

1. When a Direct Matrix Abstraction Program (DMAP) is not used, the solution is recommended and the subset associated with a solution is optional. (Default is 1,0.)
2. For Displacement Approach Rigid Formats, APP DISP, the integer value for K1 or the alphabetic characters for A must be selected from the following table:

K1	A	Description
1	STATICS or STATIC (Default)	Static analysis
2	INERTIA	Static analysis with inertia relief
3	MODES or NORMAL or REALEIGS	Real eigenvalue analysis
4	DIFFER	Static analysis with differential stiffness
5	BUCKLING	Static buckling analysis
6	PIECEWISE	Piece wise linear analysis
7	DCEIGS	Direct complex eigenvalue analysis
8	DFREQ	Direct frequency response analysis
9	DTRANS	Direct transient analysis
10	MCEIGS	Modal complex eigenvalue analysis
11	MFREQ	Modal frequency response analysis
12	MTRANS	Direct transient analysis
13	NORMALMO	Normal modes with differential stiffness
14	CYCSTATX	Static cyclic symmetry
15	CYCMODES	Real eigenvalue cyclic symmetry

3. Subsets cause a reduction in the number of statements in a Rigid Format. The use of a subset is optional.The integer value(s) may be selected from the following table:

K2, Subset	Description
1	Delete loop control.
2	Delete mode acceleration method of data recovery (modal transient and modal frequency response)
3	Combine subsets 1 and 2.

K2, Subset	Description
6	Not used.
7	Delete structure plotting and X-Y plotting.
8	Delete Grid Point Weight Generator.
9	Delete fully stressed design (static analysis).

Multiple subsets may be selected by using multiple integers separated by commas.

TIME - *Maximum Execution Time Declaration*

Description

Establishes the maximum time in minutes allotted to the execution of the NASTRAN program.

Format and Examples

```
TIME n
TIME 5
TIME 60
```

Option	Meaning
n	Integer number of minutes for execution.

Remarks

1. The default is 2^32-1 minutes.

$- COMMENT - *Comment Indicator*

Description

Declares the character string is a non-executable comment.

Format and Examples

$ Any character string°
$ COMMENTS MAY APPEAR IN ANY COLUMNS
$ SPECIAL CHARACTERS MAY BE INCLUDED ( ) + . /

Remarks

1. A comment can be added to any statement

$- COMMENT
EXECUTIVE CONTROL SECTION

2 SUBSTRUCTURAL ANALYSIS

The substructural analysis capability in NASTRAN-CORE provides a powerful capability for modeling large complex structures by assembling several component parts. For example, a complete aircraft model could be created as an assembly of major components such as wings, engine nacelles, empennage, and body. Each of these major components could in turn be assembled from more atomic components.

The part-based capability is aligned with the concept of a product definition data base. A finite element model of a component can be created using the geometric models of components and material specifications. AMSS then provides the constructive tools to assemble the components, apply loads and constraints and determine the response of the assembly to static and dynamic loads.

A part-oriented paradigm allows structural components to be duplicated or transformed. A single part can be created and assigned a name. Additional parts can then be created as transformations of primary part.

Comparing AMSS and Superelements

Superelements

The superelement approach is a technique for automatically subdividing a large model into separate components. The entire model is defined and then partitioned into smaller components that are computational more tractable. If multiple groups are involved in model generation each must be assigned a unique set of grid and element ID's and the interface grids must be exactly the same. The superelement approach is not appropriate for the development of part based assemblies.

Substructures

The design and manufacture of many complex systems involves the management of multiple vendors each tasked with the design, test and manufacture of one or more components. In the system integration design phase performance studies must be made of the assembled components. Substructuring allows each of the vendors to create structural models independently. In the assembly phase each model is encapsulated so that each component can be modeled without regard to element or grid ID's.

Basic Concepts and Terminology

Analysis Control

Substructuring is controlled by:

1. Using the SUBS subset on the APP Executive Control statement. Using the SUBS keyword, a set of DMAP statements appropriate for the selected SOL and the Substructure Control commands will be included in the standard solution sequence as an a set of ALTERs. There are no special solution sequence for AMSS.

   The AMSS system incorporates a separate data base file, called the Substructure Operating File (SOF). The SOF can be exported in either binary or character formats allowing the SOF data base to be moved between different computers. SOF files can be merged using techniques described in a later section.

2. Inserting a packet of substructure control commands after the Executive Control section. Substructures are identified by their name. Operations are then associated with a specific named substructure. The proper use of AMSS commands and the detailed form of the commands are described later in this manual.
3. Use to direct processing of one of specific substructural operational phases which are described later.
4. Substructure-specific bulk data entities.

Parts

Substructural analysis begins with the creation of one or more parts, or components. These parts are called Basic Substructures and will be identified as such on the SOF Table of Contents (SOFTOC). Each part of the structure must be created and saved in a separate NASTRAN-CORE input file. The input file associated with each part will be processed in a separate run and the associated matrices and tables will be written to the SOF using the part name as a data base qualifier, In subsequent runs secondary parts can be created as transformations, such as mirroring and rotating, or by combining two or more basic or secondary parts to create new named parts. Using the operations for combining structures multilevel subcomponents can be created and checked out in separate runs or by separate organizations.

Part Naming Conventions

Each part is a separate model which is completely independent of any other part in the structure. Grid points are assigned to a named part so they have a part qualifier in the data base. This is completely different than the superelement approach that explicitly assigns a grid point to a superelement. For this reason component names cannot be repeated. Every part must have a unique name. However, a basic part can be replicated by using the equivalence operation that replicates the part and assigns a new name to it.

Image Parts

Image Parts are created when a complex part is replicated using the **EQUIVALENCE** command. The parts that logically precede the equivalenced part are automatically created and are called image parts. This is explained in detail in a later section. Any part that is not an image part is called a primary part.

Part Connectivity Diagram

A Part Connectivity Diagram (PCD) shows the logical relation between all parts. The PCD includes the names of all parts and the procedure for creating the complete structure. The PCD is useful for configuration management of the analysis, scheduling the analysis steps, and understanding the data flow of the analysis. For example, **Figure 2-1** shows a PCD of a system model that is created from three basic parts.

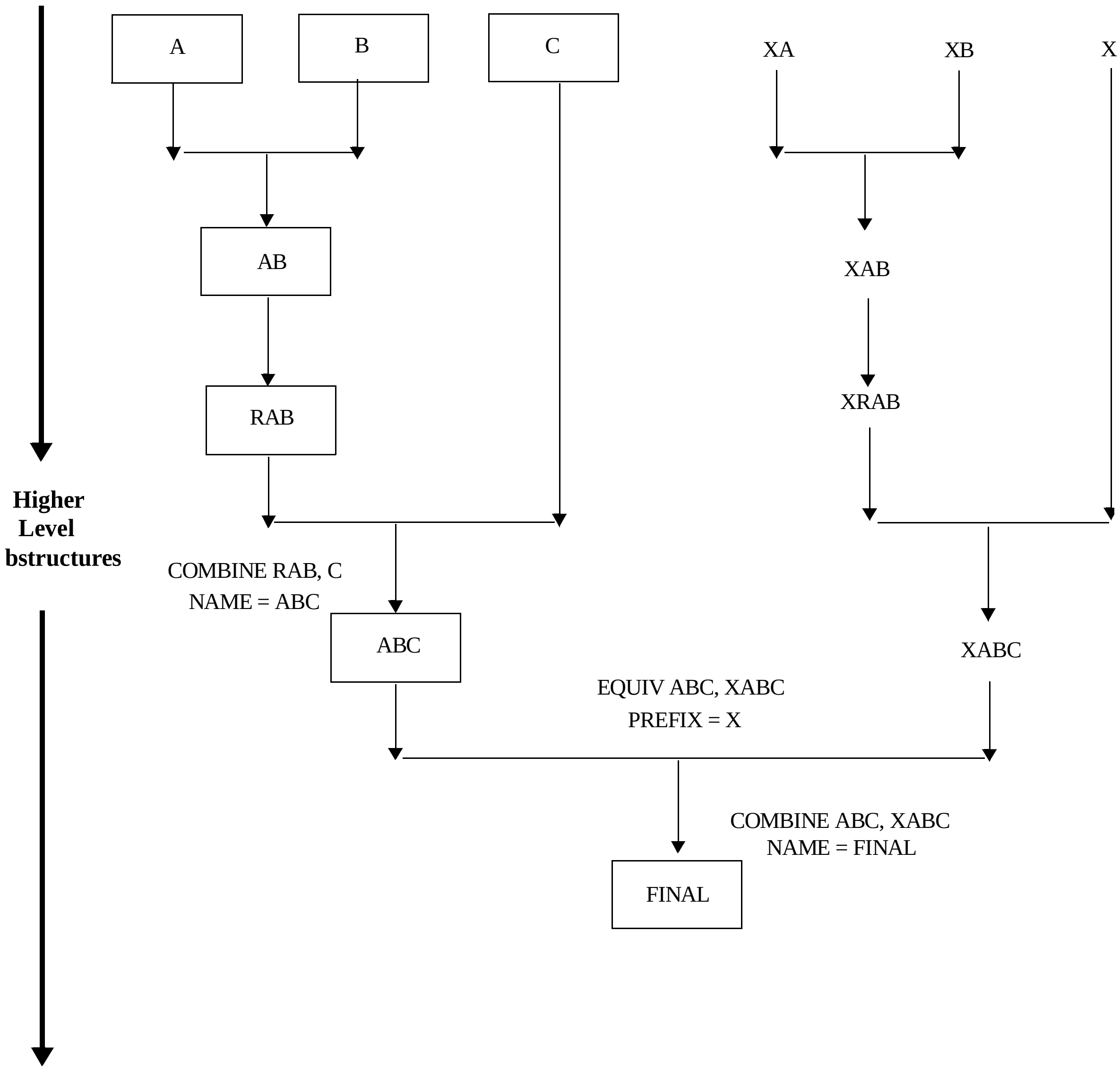

A
B
C
XA
XB
X
AB
XAB
RAB
XRAB
Higher Level bstructures
COMBINE RAB, C
NAME = ABC
ABC
XABC
EQUIV ABC, XABC
PREFIX = X
COMBINE ABC, XABC
NAME = FINAL
FINAL

Figure 2-1

Substructuring Phases

The NASTRAN-CORE substructure analysis consists of three distinct phases:

- PHASE 1 - *Part generation*. Processes the finite element model for each component and saves it on the data base
- PHASE 2 - *Assembly and solution*. Combines the parts to create the assembly and solves for the response.
- PHASE3 - *Data recover*. Recover the requested results for each of the component parts

Phase 1

A separate model and an associated execution of NASTRAN-CORE is required for each separate primary part in the assembly. Generally, permanent displacement boundary conditions and loads are defined in Phase 1. However, it is possible to modify some model characteristics, such as boundary conditions and loads, in a later phase of the substructure analyses.

Depending on the solution sequence. all appropriate matrix partitioning and reduction operations are performed to reduce the Phase 1 state to the equations of motion associated with the boundary set for the part. Therefore at the completion of Phase 1 for part i, the matrices $[K^i_{aa}]$, $[M^i_{aa}]$,$[B^i_{aa}]$ $[K^{4i}_{aa}]$,and ,$[P^i_a]$ are created. These matrices are the written to the SOF data base for later use in other substructuring operations.

The a-set are those retained during the static reduction process. In substructuring the set of a-set degrees of freedom for a part is termed the boundary dof. Any interface points between parts must be included in each part’s a-set. The a-set boundary dof must be defined since they will be used in a later **COMBINE** operation that couples two or more parts. This is done using the Case Control Command:

boundary = *sid*

The boundary sid references **BDYS** and **BDYS1** bulk data entries which define the boundary, or retained dof. All other dof are omitted.

Basic parts can be created by independent organizations that use completely different conventions for number of objects such as GRID points and elements. The most important consideration in creating the Part model is the relative orientation and position of GRID point that will connect the parts during the **COMBINE** operation. The connecting GRIDs on the parts can even be defined in different displacement coordinate systems; any differences can be resolved when the parts are combined. In addition, there are no restrictions on the use of local coordinate systems in different parts; these differences can also be resolved when the parts are combined.

Phase 2

Phase 2 operations **COMBINE**, **REDUCE** and **SOLVE** the assembly of part objects. These operations can be performed in one or more executions, each of which access the same SOF data base. Any number of **COMBINE** and **REDUCE** operations can be performed in a single execution; however, only one **SOLVE** operation can be performed in Phase 2. Initial data recovery must be performed in PHASE 2. However, the Phase 2 operations only recover the a-set for the basic Parts. Therefore a Phase 3 run must be performed for each individual part for which the part g-set, stresses and forces are required. For ease of use, all Phase 2 operations except for **SOLVE** can be performed in Phase 1.

Phase 2 operations are often used to create substructures from assemblages of parts or other reduced substructures that have been created previously. Each combine operation can be used to create a new substructure from as many as seven components. Each of the components substructures can be translated, rotated and symmetrically reflected prior to the COMBINE operation. Identical components can be created using the **EQUIVALENCE** command.

Phase 3

Phase 3 executions are used to recover element stresses and forces for all g-set dof displacements of a specified part. A separate Phase 3 run is made for each basic part for which data recovery is desired. It is also possible to recover the results for all equivalent and image components in the same Phase 3 run.

Supported Solution Algorithms

Substructure Solutions

Substructures is limited to the linear solution algorithms shown by the following table:

Table 2-1 Substructure Solutions

Type of Solution	Solution Number
Linear static analysis	1
Linear statics with inertial relief	2
Normal modes and frequencies	3
Direct frequency response	8
Modal frequency response	11
Direct transient response	9
Modal transient response	12

SOF Data Block Items

During the varios phases of substructural analysis data blocks will be mved to the Substructure Operation File data base. The items that are written to the SOF are identified in **Table 2-2**. Each of the items shown is qualified by the name of the name of the substructure component.

Table 2-2 SOF Items

Item	Description
EQSS	External grid point and internal point equivalence data.
BGSS	Basic grid point coordinates.
CSTM	Local coordinate system transformation matrices
LODS	Load set identification numbers
LOAP	Load set identification numbers for appended load vectors.
PLTS	Plot sets and other data required for Phase 2 plotting.
KMTX	Stiffness matrix.
LMTX	Decomposition product of REDUCE operation.
MMTX	Mass matrix.
PAPP	Appended load vectors
PVEC	Load vectors
POAP	Appended load vectors on omitted points
POVE	Load vectors on points omitted during matrix reduction.
UPRT	Partitioning vector used in matrix reduction.
HORG	H or G transformation matrix.
UVEC	Displacement vectors or eigenvectors.
QVEC	Reaction force vectors.
SOLN	Load factor data or eigenvalues used in a solution.
LAMS	Eigenvalue data from modal reduce operation.
PHIS	Eigenvector matrix
GIMS	G transformation matrix for interior points from a modal reduction
K4MX	Structural damping matrix.
BMTX	Viscous damping matrix.
PHIL	Left side eigenvector matrix from unsymmetric CREDUCE operation.
HLFT	Left side H transformation matrix from unsymmetric CREDUCE operation.

Substructure Control Section -

Format and Summary

The Substructure Control Section (SSCS) options provide commands needed to control the execution of NASTRAN-CORE for automated multi-stage substructure analyses (AMSS). These commands are input with the same format conventions as are used for the NASTRAN-CORE Case Control Section.

The following Executive Control Section command:

```
APP DISPLACEMENT,SUBS
```

causes the solution sequence specified by the SOL Executive Control statement to be modified to perform the operations requested in the SSCS. The standard Case Control Section, which follows the Substructure Control Section in the NASTRAN-CORE input data file, specifies the loading conditions, omit sets, method of eigenvalue extraction, element sets for plotting, plot control, output requests, etc.

The SSCS commands are summarized under one of three categories according to whether they:

- Specify the phase and mode of execution, **Table 2-3**;
- Define and control the Substructure Operating File (SOF)**Table 2-4**; or
- Specify the substructuring matrix operations, **Table 2-5**.

Several commands have associated with them a set of subcommands used to specify additional control information appropriate to the processing requested by the primary command. These subcommands are defined together with the alphabetically sorted descriptions of their primary commands in **Substructure Control Directive Descriptions** (p. 57).

The sections that follow discuss the interaction between the substructure commands and the standard case control commands, the translation of substructure commands into DMAP ALTER sequences which modify the rigid format, and the format conventions to be used. The bulk data provided for substructure analyses are included with the standard bulk data descriptions in Chapter 4 and are summarized for convenient reference in **Table 2-6**, **Table 2-7** and **Table 2-8** which describe the Bulk Data associated with REDUCE, COMBINE and SOLVE operations, respectively.

Table 2-3 Summary of Substructure Commands for Phase and Mode Control

Command	Subcommand	Description
SUBSTRUCTURE		Defines execution of substructuring phase (PHASE1, 2 or 3) (required).
	NAME*	Specifies PHASE1 substructure name.
	SAVEPLOT	Requests save of plot data for PHASE 1.
OPTIONS		Defines matrix operations (K, B, M, K4, P or PA).
RUN		Limits modes of execution (DRY, GO, DRYGO, STEP).
ENDSUBS		Terminates substructure control section (required).

*.Require name of part

Table 2-4 Summary of SOF Controls

Command	Subcommand	Description
SOF		Assigns physical files for storage of the SOF (required).
PASSWORD		Protects and ensures access to correct file SOFOUT or SOFIN Copies SOF data to or from an external file.
SOFOUT or SOFIN		Copies SOF data to or from an external file.
	POSITION	Specifies initial position of input file.
	NAMES	Specifies substructure name used for input.

Table 2-4 Summary of SOF Controls (Continued)

Command	Subcommand	Description
	ITEMS	Specifies data items to be copied in or out.
SOFPRINT		Prints selected items from the SOF.
DUMP		Dumps entire SOF to a backup file.
RESTORE		Restores entire SOF from a previous DUMP operation.
CHECK		Checks contents of external file created by SOFOUT.
DELETE		Edits out selected groups of items from the SOF.
EDIT		Edits out selected groups of items from the SOF.
DESTROY		Destroys all data for a named substructure and all the substructures of which it is a component.

Table 2-5 Summary of Substructure Operations

Command	Subcommand	Description
COMBINE		Combines sets of substructures.
	NAME*	Names the resulting substructure.
	TOLERANCE*	Limits distance between automatically connected grids.
	CONNECT	Defines sets for manually connected grids and releases.
	OUTPUT	Specifies optional output results.
	COMPONENT	Identifies component substructure for special processing.
	TRANSFORM	Defines transformations for named component substructures.
	SYMTRANSFORM	Specifies symmetry transformation.
	SEARCH	Limits search for automatic connects.
EQUIV		Creates a new equivalent substructure.
	PREFIX*	Prefix to rename equivalenced lower level substructures.
REDUCE		Reduces substructure matrices.
	NAME*	Names the resulting substructure.
	BOUNDARY*	Defines set of retained degrees of freedom.
	OUTPUT	Specifies optional output requests.
	RSAVE	Indicates the decomposition product of the interior point stiffness matrix is to be stored on the SOF.
MREDUCE		Reduces substructure matrices.
	NAME*	Names the resulting substructure.
	BOUNDARY*	Defines set of retained degrees of freedom.
	FIXED	Defines set of constrained degrees of freedom for modes calculation.
	RNAME	Specifies basic substructure to define reference point for inertia relief shapes.
	RGRID	Specifies grid point in the basic substructure to define reference point for inertia relief shapes. Defaults to origin of basic substructure coordinate system.
	METHOD	Identifies EIGR or EIGRL Bulk Data entry.
	RANGE	Identifies frequency range for retained modal coordinates.
	NMAX	Identifies number of lowest frequency modes for retained modal coordinates.
	OLDMODES	Flag to identify re-running problem with previously computed modal data.

Table 2-5 Summary of Substructure Operations (Continued)

Command	Subcommand	Description
	OLDBOUND	Flag to identify re-running problem with previously defined boundary set.
	USERMODES	Flag to indicate modal data have been input on bulk data.
	OUTPUT	Specifies optional output requests.
	RSAVE	Indicates the decomposition product of the interior point stiffness matrix is to be stored on the SOF.
CREDUCE		Reduces substructure matrices using a complex modes transformation.
	NAME*	Names the resulting substructure.
	BOUNDARY*	Defines set of retained degrees of freedom.
	FIXED	Defines set of constrained degrees of freedom for modes calculation.
	METHOD	Identifies EIGC Bulk Data entry.
	RANGE	Identifies frequency range of imaginary part of the root for retained modal coordinates.
	NMAX	Identifies number of lowest frequency modes for retained modal coordinates.
	OLDMODES	Flag to identify re-running problem with previously computed modal data.
	RSAVE	Indicates the decomposition product of the interior point stiffness matrix is to be stored on the SOF.
MRECOVER		Recovers mode shape data from an MREDUCE or CREDUCE operation.
	SAVE	Stores modal data on SOF.
	PRINT	Stores modal data and prints data requested.
SOLVE		Initiates substructure solution (statics, normal modes, frequency response or transient analysis).
RECOVER		Recovers Phase 2 solution data.
	SAVE	Stores solution data on SOF.
	PRINT	Stores solution and prints data requested.
	DISP	Displacement output request.
	SPCF	Reaction force output request.
	OLOAD	Applied load output request.
	VELO	Velocity output requests.
	ACCE	Acceleration output requests.
	BASIC	Basic substructure for output requests.
	SORT	Output sort order.
	SUBCASES	Subcase output request.
	MODES	Modes output request.
	RANGE	Mode range output request.
	ENERGY	Modal energies output requests.
	UIMPROVE	Improved displacement request.
	STEPS	Frequency or time step output request.
BRECOVER		Basic Substructure data recovery, Phase 3.
PLOT		Initiates substructure undeformed plots.

*.

Table 2-6 Summary of Substructure Bulk Data Statements for Processing REDUCE, MREDUCE and CREDUCE

Bulk Data Name	Description
BDYC	Combination of substructure boundary sets of retained degrees of freedom or fixed degrees of freedom for modes calculation.
BDYS	Boundary set definition.
BDYS1	Alternate boundary set definition.

Substructure Bulk Data -

Summary and Use

Bulk Data Used for Processing Substructure Commands REDUCE, MREDUCE, and CREDUCE

Table 2-7 Summary of Substructure Bulk Data Statements for Processing COMBINE

Bulk Data Name	Description
CONCT	Specifies grid points and degrees of freedom for manually specified connectivities - will be overridden by RELES data.
CONCT1	Alternate specification of connectivities.
RELES	Specifies grid point degrees of freedom to be disconnected - overrides CONCT and automatic connectivities.
GTRAN	Redefines the output coordinate system grid point displacement sets.
TRANS	Specifies coordinate systems for substructure and grid point transformations.

Table 2-8 Summary of Substructure Bulk Data Statements for Processing SOLVE

Bulk Data Name	Description
LOADC	Defines loading conditions for static analysis.
MPCS	Specifies multipoint constraints.
SPCS	Specifies single point constraints.
SPCS1	Alternate specification of single point constraints.
SPCSD	Specifies enforced displacements for single point constraints.
DAREAS	Defines dynamic load scale factors.
DELAYS	Defines dynamic load time delays.
DPHASES	Defines dynamic load Phase leads.
TICS	Defines transient initial conditions.

Commands and Their Execution

The sequence of operations is controlled by the order in which NASTRAN-CORE encounters the sub-structure commands. A few special commands are required in any Substructure Command Section. These are:

SUBSTRUCTURE [PHASE 1 / PHASE 2 / PHASE 3]	Required first statement in the substructure command section. It follows the CEND statement in the Executive Control Section.
SOF **PASSWORD**	Required to define the substructure operating file to be used for the current problem execution.
ENDSUBS	Specifies the end of the SSCS.

The first step of any substructuring analysis is to define the basic substructures to be used. These are prepared by executing one Phase 1 run for each substructure. Checkpoints may be taken for each Phase 1 execution to save the files that will be used during the Phase 3 data recovery runs. Alternatively, the entire original data file can be resubmitted for a Phase 3 run, thereby avoiding a proliferation of checkpoint files.

During a Phase 2 execution, a long list of instructions may be specified. This list may be split up and run in several separate smaller steps. No checkpointing is required during a Phase 2 run since all pertinent substructure data will be retained on the substructure operating file (SOF).

The Case Control Section submitted following SSCS will be used to direct the processing appropriate to the particular phase being executed. During a Phase 1 run Case Control specifies the loading conditions, single and multipoint constraints (only one set may be used per basic substructure), omits, and desired plot sets. During a Phase 2 run Case Control specifies the loads and constraint data for the SOLVE operation, outputting of results, or any plot requests. Finally, for a Phase 3 execution Case Control specifies the output and plot requests for each basic substructure.

Normal substructuring analyses will require the execution of many steps for Phase 2 processing. They may all be submitted for processing as a sequence of runs, or they may be divided into several shorter sequences and executed separately. If there is an abnormal termination, several steps may have been successfully executed. To recover requires simply removing those completed steps from the SSCS and re-submitting the remaining directives. The SOF will act as the checkpoint/restart file independently of the normal NASTRAN-CORE checkpointing procedures.

If the solution structure is large, a NASTRAN-CORE checkpoint is recommended to save intermediate results during the SOLVE operation. If this is done, however, care must be exercised on restart to insure correct re-entry into the DMAP sequence. This may be accomplished by removing all substructure control commands preceding the SOLVE, modifying the Case Control and Bulk Data sections to change set identifiers only if any new loads or constraint sets are to be specified, and re-submitting the run. If no changes are to be made affecting the SOLVE operations, a regular restart can be executed without changing the original Case Control and Bulk Data sections.

Interface to DMAP -

DMAP Alters

Each substructure command generates a set of DMAP ALTER statements which are automatically inserted into the Solution sequence associated with the SOL directive of the Execution Control section. These ALTER's require no user intervention except as follows:

1. User specified ALTER's can be included in Executive Control. However, they may not overlap any DMAP statements affected by the substructure ALTERs.
2. The AMSS DMAP can be suppressed and included as explicit ALTER statement or by using approach DMAP. To suppress the automatic ALTER generation, the following forms of the executive control APP directive are provided.

   APP DISP,SUBS,1 (Retains execution of the substructuring preface operations.
   or
   APP DMAP (Standard NASTRAN-CORE is executed)
3. For user information and convenience, the substructure ALTER packages may be printed and/or written to the system PUNCH file. The inclusion of DIAG 23 in Executive Control will produce the printout. DIAG 24 will produce the punch file. The punch file can be modified and resubmitted as described above. However, the order of the associated SSCS must not be changed, to insure proper sequencing of the requested operations.

Substructure Control Directive Descriptions

The format of the Substructure Control directives is free-field. Blanks are used to separate the control words. Either a blank or an equal sign (=) can be used in an assignment statement. Comments, at the beginning of an input line, can be inserted anywhere in the SSCS and may contain any alphanumeric characters. Only the first four characters of each directive need be used so long as that option is uniquely identified.

1. In presenting general formats for each directive embodying all options, the following conventions are used:
2. Upper-case letters and parentheses must be input as shown.
3. Lower-case letters indicate that a substitution must be made.
4. Double brackets $[[Mandatory]]$ indicate that a choice of contents is mandatory.
5. Brackets $[Optional]$ contain an option that may be omitted or included by you.
6. First listed options or values are the default values.
7. Physical record consists of information in characters in 1 through 72. All Substructure Control commands are limited to a single physical record.

The Case Control Section follows the ENDSUBS command of the Substructure Control Section.

BRECOVER - *Basic Substructure Data Recovery*

Purpose

This operation is performed in Phase 3 to recover detailed output data for a basic substructure used in Phase 1.

Request Format

```
BRECOVER name
```

Subcommands

None.

Definitions

Item	Description
name	Name of structure defined in Phase 1 or structure equivalenced to the Phase 1 structure.

Remarks

1. Use of the RECOVER command in Phase 3 has the same effect as BRECOVER. That is, RECOVER is an alias for BRECOVER in Phase 3.
2. Phase 3 may be a RESTART of the original Phase 1 run or it may be executed from the original input data.

CHECK - *Check Contents of External File*

Purpose

To list all substructure items on an external file which was generated with SOFOUT.

Request Format

CHECK filename, $\left[\begin{bmatrix}\text{FILE}\\\text{DISK}\end{bmatrix}\right]$

Subcommands

None.

Definitions

Item	Description
filename	Name of the external file. One of the following: INPT, INP1,..., INP9.
TAPE	File resides on sequential access device.
DISK	File resides on a direct access device.

Remarks

1. The substructure name, item name, and the date and time the item was written are listed for each item on the file.

COMBINE - *Combine Sets of Substructures*

Purpose

This command will perform the operations to combine the matrices and load up to seven substructures into matrices and loads representing a new pseudo structure. Each component structure may be translated, rotated, and reflected before it is connected. You may manually select the points to be connected or direct the program to connect them automatically.

Request Format

$$\text{COMBINE}\left(\begin{bmatrix}\text{AUTO}\\ \text{MAN}\end{bmatrix}, \begin{bmatrix}X\\ Y\\ Z\end{bmatrix}\right)\text{name1, name2, etc.}$$

Subcommands

```
NAME       new name (required)
TOLERANCE  i (required)
CONNECT    n
OUTPUT     m1, m2,...
```

Each individual *component* substructure may have the following added commands:

```
COMPONENT = name
TRANSFORM = m
```

$$\text{SYMTRANSFORM} = \begin{bmatrix}X\\ Y\\ Z\\ XY\\ YZ\\ ZX\\ XYZ\end{bmatrix}$$

```
SEARCH = namej, namek, etc.
```

Definitions

Command	Description
AUTO/MAN	Defines method of connecting points. If AUTO is chosen, the physical location of grid points is used to automatically determine connections. If MAN is chosen, all connections must be manually defined on CONCT or CONCT1 bulk data entries.
X, Y,Z	Are used on COMBINE directive for searching geometry data for AUTO connections. Denotes preferred search direction for processing efficiency. See Remark 1.
name1, name2, etc.	Unique names of substructures to be combined. Limit is from one to seven component structures. See Remarks 5 and 6.
new name	Defines name of combination structure (required).
i	Defines limit of distance between points which will be automatically connected (real > 0).

Command	Description
n	Defines set number of manual connections and releases specified on bulk data entries, CONCT, CONCT1, and RELES.
name	On COMPONENT entry, defines the substructure (name1, etc.) to which the subsequent data is applied.
m	Set identification number of TRANS and GTRAN bulk data entries which define the orientation of the substructure and/or selected grid points relative to new basic coordinates. See Remarks 2 and 3.
X,Y,...XY,...XYZ	Defines axis (or set of axes) normal to the plane(s) of symmetry in the new basic coordinate system. The displacement and location coordinates in these directions will be reversed in sign. See Remarks 2 and 3.
namej	Limits the automatic connection process such that only connections between component "name" and these structures are produced. Multiple search commands may appear for any one component. See Remark 4.
m1, m2, etc.	Optional output requests. See Remark 7.

Remarks

1. The automatic connections are produced by first sorting the grid point coordinates in the specified coordinate direction and then searching within limited groups of coordinates. If the boundary of a substructure to be connected is aligned primarily along one of the coordinate axes, this axis should be used as the preferred search direction. If the boundary is parallel with, say, the yz plane and all boundary coordinates have a constant x value, then the search should be specified along either the y or the z axis.
2. The transformation (TRANS) data defines the orientation of the component substructure (old basic) in terms of the new basic coordinate system. All grid points originally defined in the old basic system will be transformed to the new basic system. Points defined in local coordinate systems will not be transformed unless otherwise specified on a GTRAN directive, and their directions will rotate with the substructure.
3. The SYMTRANSFORM (or SYMT) directive is primarily used to produce symmetric reflections of a structure. This is usually preceded by an EQUIV directive to produce a new, unique substructure name. Note that the results for the new reflected substructure may reference a left-handed coordinate system wherever local coordinate systems are retained during the transformation. However, those coordinates which are originally in the old basic or are newly specified via a GTRAN directive are automatically transformed to a right-handed coordinate system of the combined structure during the combination process. Note that the symmetric reflection occurs first using the component's own basic coordinate system before the translational and rotational transformation called for by TRANS.
4. If any search option is present, then all connections between substructures must be specified explicitly with SEARCH commands. Only those combinations specified will be searched for possible connects. Symmetric connects need not be declared (that is, COMPONENT A SEARCH B implies COMPONENT B SEARCH A). Care must be taken to assure all proper connections of substructures should any SEARCH commands be utilized.
5. Matrix data for the COMBINE operation is automatically processed in the most economical order, that is, the matrices with fewest terms are processed first.
6. The bandwidth of the resultant matrices may be controlled by selection of substructures, their boundaries, and the order in which the substructures are listed in the COMBINE command. The degrees of freedom in the resultant matrices are located as defined in the sample problem below:

```
COMBINE A, B, C, D
```

Item	Description
A	Interior
AB	Boundary
B	Interior
AC	Boundary

Item	Description
ABC	Boundary
B	Interior
AD	Boundary
BC	Boundary
etc.	

The following output requests are available for the COMBINE operation.

Code Output	Description
2*	SOF table of contents.
3	CONCT1 bulk data summary.
4	CONCT bulk data summary.
6	GTRAN BD summary.
7*	TRANS BD summary.
9	RELES BD summary.
11	Summary of automatically generated connections (in terms of internal point numbers).
12*	Complete connectivity map of final combined pseudo structure defining each internal point in terms of the grid point ID and component substructure it represents.
13	The EQSS item.
14	The BGSS item.
15	The CSTM item.
16	The PLTS item.
17	The LODDS item.

*.Recommended output options.

For requests 13-17, output printed is formatted SOF data for the newly created pseudo structure. See **SOF** (p. 89) for definitions.

Examples

```
COMBINE PANEL SPAR
TOLE = .0001 NAME = SECTA

COMBINE (AUTO,Z) TANK1, TANK2, BULKHD
NAME = TANKS
TOLE = .01
COMPONENT TANK1
TRAN = 4
SEARCH = BULKHD
COMPONENT TANK2
SEARCH = BULKHD

COMBINE (MAN) LWING, RWING
```

```
TOLE = 1.0
NAME = WING
COMPONENT LWING
SYMT = Y
```

CREDUCE - *Reduces Substructure Matrices Using Complex Modes*

Purpose

This command performs a complex modal synthesis reduction on a specified component substructure. The resulting substructure will be defined by boundary point displacements and modal displacements as degrees of freedom. The operation is allowed in both Phase 1 and Phase 2 jobs and may be performed at any level of the substructure process.

Request Format

```
CREDUCE name
```

Subcommands

```
NAME       new name (required)
BOUNDARY   b (required)
FIXED      f
METHOD     k
RANGE      f1, f2
NMAX       N
OUTPUT     m1, m2
OLDMODES   m
GPARAM     g
RSAVE      (See Remark 4)
```

Definitions

Command	Description
name	Name of substructure to be reduced.
new name	Name of resulting structure.
b	Set identification number of a BDYC Bulk data sttement which define sets of boundary degrees of freedom (Integer > 0). See remark 1.
f	Optionally identifies BDYC entry defining degrees of freedom temporarily fixed during mode extraction (Integer ≥ 0, default = 0).
k	Identifies EIGC bulk data entry for control of the eigenvalue extraction (Integer > 0).
f1,f2	Optional frequency range (Hz) for the imaginary part of the root defining eigenvectors to be used in the mode synthesis formulation (Real, default = ALL).
N	Optional number of lowest modes, measured by magnitude of eigenvalue, within frequency range to be used in mode synthesis formulation (Integer, default = ALL).
m1,m2	Optional output requests.
m	Flag for re-running problem with old eigenvectors.
g	Structural damping coefficient.

Remarks

1. All references to the grid points and components not defined in the "boundary set" will be reduced out of the new substructure. Any subsequent reference to these omitted degrees of freedom in COMBINE, CREDUCE, or SOLVE operations generates an error condition.
2. The following output requests are available for the CREDUCE operation:

Code Output	Description
1*	Current problem summary.
2	Boundary set summary.
3	Summary of grid point ID numbers in each boundary set.
4	The EQSS item for the substructure being reduced.
5*	The EQSS item.
6*	The BGSS item.
7	The CSTM item.
8	The PLTS item.
9*	The LODS item.
10*	Modal dof set summary.
11	Fixed set summary.
12	Summary of grid point ID numbers for the new reduced pseudo structure.

*.Recommended output options.

Requests 5-8 write formatted SOF items for the new reduced pseudo structure.

3. The OLDMODES option instructs the program to use the existing modal data but create new boundary matrices for a new boundary set. To exercise the OLDMODES option, you must use the following sequence of commands to eliminate previously calculated boundary point data:

```
EDIT(32) new name (previous modal reduction name)
DELETE name, GIMS, LMTX, HLFT, HORG, UPRT
DELETE name, POVE, POAP
CREDUCE name : :
```

4. If the RSAVE statement is included, the decomposition product of the interior point stiffness matrix (LMTX item) is saved on the SOF file. This matrix will be used in the data recovery for the omitted points. If it is not saved, it will be regenerated when needed.

DELETE - *Delete Items from SOF*

Purpose

To delete individual substructure items from the SOF.

Request Format

```
DELETE name, item1, item2, item3, item4, item5
```

Subcommands

None.

Definitions

Name	Description
name	Substructure name
item1, item2	Item names (HORG, KMTR, LODS, SOLN, etc.)

Remarks

1. DELETE may be used to remove from one to five items of any single substructure.
2. For primary substructures, items of related secondary substructures are removed only if the latter point to the same data (KMTX, MMTX, etc.).
3. For secondary and image substructures, no action is taken on items of related substructures, that is, items of equivalenced substructures or higher or lower level substructures.
4. See the **DESTROY** (p. 67) and **EDIT** (p. 69) commands for other means of removing substructure data.

DESTROY - *Removes All Data Referencing a Component Substructure*

Purpose

To remove data for a substructure and all substructures of which it is a component from the SOF. In addition to the substructure being DESTROYed ("name"), data for substructures which satisfy one or more of the following conditions are also removed from the SOF:

1. All substructures of which "name" is a component
2. All secondary (or equivalenced) substructures for which "name" is the primary substructure
3. All image substructures which are components of a substructure that is destroyed

Request Format

```
DESTROY name
```

Subcommands

None.

Definitions

Item	Description
name	Name of substructure

Remarks

1. No action is taken if "name" is an image substructure.
2. See related commands EDIT and DELETE for additional means of removing substructure data.

DUMP - *Copy SOF to External File*

Purpose

To copy the entire SOF to an external file.

Request Format

$$\text{DUMP Filename,} \left[\begin{bmatrix} \text{FILE} \\ \text{DISK} \end{bmatrix} \right]$$

Subcommands

None.

Definitions

Item	Description
filename	Name of external file. Anyone of the following: INP1, INP2, ... , INP9.
TAPE	File is a sequential access device
DISK	File is a direct access device.

Remarks

1. DUMP may be used to create a backup copy of the SOF.
2. All system information on the SOF is saved.
3. The RESTORE command will reload a DUMPed SOF.
4. DUMP/RESTORE may not be used to change the size of the SOF.
5. It is more efficient to use operating system utility programs, if available, to create back-up copies of the SOF.

EDIT - *Selectively Removes Data from SOF File*

Purpose

To permanently remove selected substructure data from the SOF.

Request Format

```
EDIT (opt) name
```

Subcommands

None.

Definitions

Item	Description
name	Name of Substructure.
opt	Integer value reflecting combinations of requests. The sum of the following integers defines the combination of data items to be removed from the SOF.

opt Value	Items Removed
1	Stiffness matrix (KMTX).
2	Mass matrix (MMTX).
4	Load data (LODS, LOAP, PVEC, PAPP).
8	Solution data (UVEC, QVEC, SOLN).
16	Transformation matrices defining next level (HORG, UPRT, POVE, POAP, LMTX, GIMS, HLFT).
32	All items for the substructure.
64	Appended loads data (LOAP, PAPP, POAP).
128	Damping matrices (K4MX, BMTX).
256	Modal reduction data (LAMS, PHIS, PHIL).

Remarks

1. You are cautioned on the removal of the transformation matrix data. These matrices are required for the recovery of the solution results.
2. For primary substructures, items of related secondary substructures are removed only if they point to the same data (KMTX, MMTX, etc.).
3. For secondary and image substructures, no action is taken on items of related substructures, that is, items of equivalenced or higher or lower level substructures.
4. If the EDIT feature is to be employed, you should consider also using SOFOUT to ensure the existence of backup data if there is an error.
5. See **DELETE** (p. 66) and **DESTROY** (p. 67) for other means of removing substructure data.

ENDSUBS - *Defines the End of the Substructure Control Deck.*

Purpose

This command terminates the processing of automated substructuring controls and directives.

Request Format

```
ENDSUBS
```

Subcommands

None.

EQUIV - *Create a New Equivalent Substructure*

Purpose

To assign an alias to an existing substructure and thereby create a new equivalent substructure. The new secondary substructure may be referenced independently of the original primary substructure in subsequent substructure commands. However, the data actually used in substructuring operations is that of the primary substructure.

Request Format

```
EQUIV name1, name2
```

Subcommands

```
PREFIX    p (required)
```

Definitions

Item	Description
p	Single character
name1	Existing primary substructure name.
name2	New equivalent substructure name.

Remarks

1. A substructure created by this directive is referred to as a secondary substructure.
2. All substructures which were used to produce the primary substructure will produce equivalent image substructures. The new image substructure names will have the prefix p.
3. A DESTROY operation on the primary substructure data will also destroy the secondary substructure data and all image substructures.
4. An EDIT or DELETE operation on the primary substructure will not remove data of the secondary substructure and vice versa.

MRECOVER - *Eigenvector Recovery for Modal Synthesis Operations*

Purpose

This command recovers modal displacements and boundary forces for substructures reduced to modal coordinates. The results are saved on the SOF file and they may be printed upon your request. This command may be input after the MREDUCE or CREDUCE commands or at a later time as desired.

Request Format

```
MRECOVER s-name
```

Subcommands

(

SAVE	cname1
PRINT	cname2
RANGE	f1, f2 (see Remark 7)
UIMPROVE	(see Remark 9)
DISP	NONE
	n
	ALL
SPCF	NONE
	n
	ALL
ENERGY	NONE
see remark 4	n
	ALL
SORT	MODES
see remark 10	SUBSTRUCTURE
BASIC	b-name
MODES	NONE
see remark 7	n
	ALL

Definitions

Item	Description
a-name	Name of the substructure that was reduced in a prior MREDUCE or CREDUCE command for which the solution results are to be recovered.
cname1	Name of the component substructure for which the results are to be recovered and saved on the SOF. May be the same as "s-name". See Remarks 1, 2, and 3.
cname2	Name of the component substructure for which the results are to be recovered and printed on the SOF. May be the same as "s-name". See Remarks 1, 2, 3, 8, and 11.
b-name	Name of the component basic substructure for which the subsequent output requests are to apply.
ALL	Output for all points will be produced. See Remark 8.

Item	Description
NONE	No output is to be produced.
n	Set identification number of a SET command appearing in Case Control. Only output for those points whose identification numbers appear on this SET command will be produced. See Remark 5.
f1, f2	Range of frequencies for which output will be produced. If only f1 is present, the range is assumed to be 0 - f1. See Remark 7.

Output Requests

Printed output produced by the MRECOVER PRINT command can be controlled by requests present in either Case Control or the MRECOVER command in the Substructure Control Deck. If no output requests are present, the PRINT command is equivalent to SAVE and no output will be printed.

The output options described above may appear after any PRINT command. These output requests will then override any Case Control requests. The output requests for any PRINT command can also be specified for any or all basic component substructures of the results being recovered. These requests will then override any requests in Case Control or after the PRINT command.

Example of output control:

```
MRECOVER SOLSTRCT
     PRINT ABDC
        SORT= SUBSTRUCTURE
        DISP = ALL
        BASIC A
           DISP = 5
        BASIC C
           SPCF = 20
     SAVE ABC
```

Remarks

1. SAVE will save the solution for substructure "name" on the SOF. PRINT will save and print the solution.
2. If the solution data already exists on the SOF, the existing data can be printed without costs of regeneration with the PRINT command.
3. For efficiency, you should order multiple SAVE and/or PRINT commands so as to trace one branch at a time starting from your solution structure.
4. Reaction forces are computed for a substructure only if (1) the substructure is named on a PRINT subcommand and (2) an output request for SPCFORCE or modal energies exists in the Case Control or the RECOVER command.
5. All set definitions should appear in Case Control to ensure their availability to the MRECOVER module.
6. The SORT output option should only appear after a PRINT command. Any SORT commands appearing after a BASIC command will be ignored.

   SORT = MODES (the default) will cause all output requests for each mode to appear together.

   SORT = SUBSTRUCTURE will cause all output requests for each basic substructure to appear together.
7. If both a MODES request and a RANGE request appear for dynamic analysis, both requests must be satisfied for any output to be produced.
8. The media, print or punch, where output is produced is controlled through Case Control requests. If no Case Control requests are present, the default of print is used.
9. If the UIMPROVE request is present for a substructure that was input to a REDUCE, MREDUCE, or CREDUCE, an improved displacement vector will be generated. This vector will contain the effects of inertia and damping forces.

10. The ENERGY request will cause the calculation of modal energies on all included and excluded modal dof for a modal reduced substructure. This request should appear for the substructure that was input to the modal reduce operation so that required data needed for the excluded mode calculations exists. This request requires that the UVEC item exist for the next higher level structure.

11. You can specify print thresholds for all printout. If the absolute value is less than the threshold, the value will be set to zero. The following thresholds can be input on PARAM bulk data entries.

Threshold	Description
UTHRESH	Displacement, velocity, and acceleration threshold.
PTHRESH	Load threshold.
QTHRESH	Reaction force threshold.

12. Since the subcommands of the MRECOVER command are all associated with a component structure, multiple use of these subcommands is permitted.

MREDUCE - *Reduces Substructure Matrices Using Real, Normal Modes*

Purpose

This command performs a modal synthesis reduction on a specified component substructure. The resulting substructure will be defined by boundary coordinate displacements and modal coordinate displacements as degrees of freedom. The operation is allowed in both Phase 1 and Phase 2 jobs and may be performed at any level of the substructure process.

Request Format

```
MREDUCE name
```

Subcommands

```
NAME       new name (required)
BOUNDARY   b (required)
FIXED      f
METHOD     k
RANGE      f1, f2
NMAX       N
RGRID      i (see Remark 12)
RNAME      c-name
RSAVE      (see Remark 7)
OLDMODES   m
OLDBOUND   n
USERMODES  j
OUTPUT     m1, m2
```

Definitions

Item	Description
name	Name of substructure to be reduced.
new name	Name of resulting substructure. See Remarks 2 and 3.
b	Set identification number of BDYC Bulk Data entries which define sets of boundary degrees of freedom (Integer). See Remark 1.
f	Optionally identifies BDYC data defining degrees of freedom temporarily fixed during mode extraction (Integer, default = 0).
k	Identifies EIGR or EIGRL Bulk Data entry for control of the mode extraction (Integer > 0).
i	Grid point number for defining origin of free body motion. Used with RNAME to define substructure component containing grid point i (Integer ≥ 0, default = 0). (See Remark 12.)
c-name	Name of basic substructure which contains grid point i. If RGRID = 0 or is missing, the origin of the overall basic coordinate system is used to define the six rigid body motions. These motions define the inertia relief deflection shapes which are used as generalized coordinates in addition to the modal coordinates.
m	Flag for re-running problem with old mode shapes (YES or NO). See Remarks 5, 8, and 10.

Item	Description
n	Flag for re-running problem with old boundaries for different eigenvalue method (YES or NO). See Remarks 5, 9, and 10.
f1, f2	Optional frequency range (in cycles per unit time) defining modes to be used in the mode synthesis formulation (Real, default = ALL).
N	Optional number of lowest modes within elastic frequency range to be used in mode synthesis formulation (Integer, default = ALL). Rigid body modes are automatically included, in addition to the selected number of NMAX of elastic modes.
j	Option used in Phase 1 when METHOD data is missing and user-input modes are used directly. See Remark 6.
m1, m2	Optional output requests. See Remark 4.

Remarks

1. All references to the grid points and components not defined in the "boundary set" will be reduced out of the new substructure. Any subsequent reference to these omitted degrees of freedom in COMBINE, MREDUCE, REDUCE, or SOLVE operations generates an error condition.
2. The resulting substructure will be defined in terms of the following degrees of freedom:

ub	Boundary grid point displacements.
ε_j	Modal displacements relative to static deflection shapes induced by boundary inertia.
ε_o	Inertia relief generalized coordinates defined by inertia relief deflection shapes occurring from boundary point rigid body accelerations (zero frequency modes).

   Note that a new substructure will be automatically created to define coordinates ε_o and ε_j. The name will be the same as given by NAME and the point identification numbers are 1-6 for ε_o and 101, 102,... for ε_j.
3. The same transformations applied to the stiffness matrix will be applied to the loads, mass, and damping matrices for the new substructure. See the NASTRAN Theoretical Manual for a discussion of this effect.
4. The following output requests are available for the MREDUCE operation:

Code	Output
1	Current problem summary.
2	Boundary set summary.
3	Summary of Grid point ID numbers for each boundary set.
4	The EQSS item for the structure being reduced.
5	The EQSS item.
6	The BGSS item.
7	The CSTM item.
8	The PLTS item.
9	The LODS item.
10	Modal dof set summary.
11	Fixed set summary.
12	Summary of Grid point ID numbers in each fixed set.

   Requests 5-9 write formatted SOF items for the new reduced pseudo structure.
5. The options OLDMODES and OLDBOUND allow you to re-run the reduction and:
   - Change the boundary without recalculating modes.
   - Change the modes without the boundary condensation calculations.
   - Select a different mode range from the existing vectors and avoid recalculating modes and boundary matrices.
6. You must provide the actual mode data in Phase 1 when USERMODES = j is given. Two options are provided:

- If j = 1, the structure must be entirely defined by a finite element model and the eigenvectors for the NASTRAN-CORE u_a set provided in data block PHIS input using DMI entries.
- If j = 2, the entire structure need not be defined. You provide eigenvectors and forces of constraint only at the selected boundary points as well as eigenvalues and modal masses. Residual stiffness and mass matrices may also be provided to define properties at the boundary points. Use DMI and DTI entries for these data.

7. If the RSAVE directive is included, the decomposition product of the interior point stiffness matrix (LMTX item) is saved on the SOF file. This matrix will be used in the data recovery for the omitted points. If it is not saved it will be regenerated when needed.
8. Exercising the OLDMODES option, you must use the following sequence of commands:

```
EDIT(32)new name (previous modal reduction name)
EDIT(16)name
MREDUCE name
NAME = new name
```

9. Exercising the OLDBOUND option, you must use the following sequence of commands:

```
EDIT(32)new name (previous modal reduction name)
EDIT(768) name
MREDUCE name
NAME = new name
```

10. Exercising both the OLDMODES and OLDBOUND options concurrently you must use the following sequence of commands:

```
EDIT(32)new name (previous modal reduction name)
EDIT(512) name
MREDUCE name
NAME = new name
```

11. You are strongly urged to select code 10 for your output request. The modal dof set summary gives a good breakdown between the assignments of rigid body modes and elastic modes. The MREDUCE module sometimes overrides your specification of NMAX. This occurs when the nature of the mode is such that the 2-3 term of $[H_{gh}]$ (as defined by **Equation 27 on page 4.7-7 of the Theoretical Manual**) is zero. When this occurs NASTRAN-CORE automatically deletes the ineffective mode from the solution set. Any such omission can be verified from the printout triggered by code 10.
12. Note on RGRID: Your choice of one grid point or another for inertia relief modes does not in any way determine the net reaction forces, but operates solely as a convenience as to choice of reference origin.

OPTIONS - *Defines Matrix Types*

Purpose

This allows you to selectively control the type of matrices being processed.

Request Format

```
OPTIONS m1,m2,m3
```

Subcommands

None.

Definitions

Item	Description
m1,m2,m3	Any combination of the characters K, M, B, K4, and either P or PA, where: K = Stiffness matrices. M = Mass matrices. P = Load matrices. PA = Appended load vectors. B = Viscous Damping matrices. K4 = Structure damping matrices.

Remarks

1. The default depends on the NASTRAN-CORE rigid format:

Rigid Format	Default
1- Statics	K, P
2 - Inertia Relief	K, M, P
3 - Normal Modes	K, M
8 - Frequency Response	K, M, P, B, K4
9- Transient Response	K, M, P, B, K4

2. In a Phase 1 execution, Rigid Formats 1 and 3 will provide only two of the matrices, as shown above. In Rigid Format 1, the mass matrix is not generated. In Rigid Format 3, the loads matrix is not generated. An error condition will result unless you add the required DMAP alters to provide the requested data.
3. Stiffness, mass, load, or damping matrices must exist if the corresponding K, M, P, PA, B, or K4 option is requested in the subsequent Phase 2 run.
4. Matrices or loads may be modified by re-running the substructure sequence for only the desired type. However, the old data must be deleted first with the EDIT or DELETE command.
5. The append load option, PA, is used when additional load sets are required for solution, and it is not desired to regenerate existing loads. To generate these new load vectors, re-execute all required Phase 1 runs with the new load sets and OPTION = PA. Then, repeat the Phase 2 operations with OPTION = PA. At each step, the new vectors are appended to the existing loads so that all load vectors will be available in the SOLVE stage.
6. Each OPTION command overrides the preceding command to control subsequent steps of the substructure process.
7. When executing the SOLVE command, the option selected must provide the matrices required for the rigid format being executed.

PASSWORD - *Substructure Operating File Declaration*

Purpose

This declaration is required in the substructure command deck. The password is written on the SOF file and is used to protect the file and ensure that the correct file is assigned for the current run.

Request Format

```
PASSWORD password
```

Subcommands

None.

Definitions

Item	Description
password	Password for the SOF (8 characters maximum).

PLOT - *Substructure Plot Command*

Purpose

This command is used to plot the undeformed shape of a substructure which may be composed of several component substructures. This command initiates the execution of a plot at any stage of the substructure process. The actual plot commands -- origin data, etc.-- must be included in the normal case control data.

Request Format

```
PLOT name
```

Subcommands

None.

Definitions

Item	Description
name	Name of component to be plotted.

Remarks

1. This PLOT command can be used in any of the three phases. However, it is suggested that it be used only in Phase 2. In the case of Phase 1 and Phase 3 runs, any desired plots can be obtained in the usual manner by appropriate requests in the structure plotter output request packet in the Case Control Deck.
2. The set of elements to be plotted in Phase 2 consists of all the elements and grid points saved in Phase 1 for each basic substructure comprising the substructures named in the PLOT command. The set definition given in the structure plotter output request packet in the Case control Deck in Phase 2 is ignored. (Only one plot set from each basic substructure is saved in Phase 1.)
3. The structure plotter output request packet, while part of the standard Case Control Section is treated separately.

RECOVER - *Phase 2 Solution Data Recovery*

Purpose

This command recovers displacements and boundary forces on specified substructures in the Phase 2 execution. The results are saved on the SOF file and they may be printed upon your request. This command should be input after the SOLVE command to store the solution results on the SOF file.

Request Format

```
RECOVER s-name
```

Subcommands

SAVE	cname1
PRINT	cname2
DISP	NONE
	n
	ALL
SPCF	NONE
see remark 4	n
	ALL
OLOAD	NONE
see remark 11	n
	ALL
BASIC	b-name
ENERGY	NONE
see remark 10	n
	ALL

• *For static analysis only:*

SORT	SUBCASE
see remark 6	SUBSTRUCTURE
SUBCASES	NONE
	n
	ALL

• *For normal modes analysis only:*

SORT	MODES
see remark 6	SUBSTRUCTURE
MODES	NONE
see remark 7	n
	ALL
RANGE	f1, f2 (see Remark 7)

• *For dynamic analysis only:*

SORT	FREQUENCY
see remark 6	TIME
	SUBSTRUCTURE
STEPS	NONE
	n
	ALL
RANGE	f1, f2 (See Remark 7)
UIMPROVE	(See Remark 9)

Definitions

Item	Description
s-name	Name of the substructure named in a prior SOLVE command for which the solution results are to be recovered.
cname1	Name of the component substructure for which the results are to be recovered and saved on the SOF. May be the same as "s-name". See Remarks 1, 2, and 3.
cname2	Name of the component substructure for which the results are to be recovered and printed on the SOF. May be the same as "s-name". See Remarks 1, 2, 3, 8, and 12
b-name	Name of the component basic substructure for which the subsequent output requests are to apply.
ALL	Output for all points will be produced. See Remark 8.
NONE	No output is to be produced.
n	Set identification number of a SET command appearing in Case Control. Only output for those points, subcases, modes, frequencies, or time steps whose identification numbers appear on this SET command will be produced. See Remark 5.
f1, f2	Range of frequencies for which output will be produced. If only f1 is present, the range is assumed to be 0 - f1. See Remark 7.

Output Requests

Printed output produced by the RECOVER PRINT command can be controlled by requests present in either Case Control or the RECOVER command in the Substructure Control section. If no output requests are present, the PRINT command is equivalent to SAVE and no output will be printed.

The RECOVER output options described above may appear after any PRINT command. These output requests will then override any Case Control requests. The output requests for any PRINT command can also be specified for any or all basic component substructures of the results being recovered. These requests will then override any requests in Case Control or after the PRINT command.

Example of output control:

```
RECOVER SOLSTRCT
     PRINT ABDC
        SORT = SUBSTRUCTURE
        DISP = ALL
        OLOAD = 10
        BASIC A
           DISP = 5
        BASIC C
           OLOAD = NONE
           SUBCASES = 20
     SAVE ABC
```

Remarks

1. SAVE will save the solution for substructure "name" on the SOF. PRINT will save and print the solution.
2. If the solution data already exists on the SOF, the existing data can be printed without costs of regeneration with the PRINT command.
3. For efficiency, multiple SAVE and/or PRINT commands can be issued so as to trace one branch at a time starting from your solution structure.
4. Reaction forces are computed for a substructure only if (1) the substructure is named on a PRINT subcommand and (2) an output request for SPCFORCE or modal energies exists in the Case Control or the RECOVER command.
5. All set definitions should appear in Case Control to ensure their availability to the RECOVER module.
6. The SORT output option should only appear after a PRINT command. Any SORT commands appearing after a BASIC command will be ignored.

   For static analysis, SORT = SUBCASE (the default) will cause all output requests for each subcase to appear together. For normal modes analysis, SORT = MODES (the default) will cause all output requests for each mode to appear together. For dynamic analysis, SORT = FREQ (the default for frequency response) or SORT = TIME (the default for transient response) will cause all output requests for each frequency or time step, as the case may be, to appear together. In all these analyses, SORT = SUBSTRUCTURE will cause all output requests for each basic substructure to appear together.

7. If both a MODES (or STEPS) request and a RANGE request appear for dynamic analysis, both requests must be satisfied for any output to be produced.
8. The medium, print or punch, where output is produced is controlled through Case Control requests. If no Case Control requests are present, the default of print is used.
9. If the UIMPROVE request is present for a substructure that was input to a REDUCE, MREDUCE, or CREDUCE, an improved displacement vector will be generated. This vector will contain the effects of inertia and damping forces.
10. The ENERGY request will cause the calculation of modal energies on all included and excluded modal dof for a modal reduced substructure. This request should appear for the substructure that was input to the modal reduce operation so that required data needed for the excluded mode calculations exists. This request requires that the UVEC item exists for the next highest level structure.

11. For dynamic analysis, the printed loads output will include dynamic loads only for the solution substructure in the same run where the solution was obtained. For any lower level substructures or on any run after the solution, only static loads will be printed.

12. You can specify print thresholds for all printout. If the absolute value is less than the threshold, the value will be set to zero. The following thresholds can be input on PARAM bulk data entries.

Threshold	Description
UTHRESH	Displacement, velocity, and acceleration threshold.
PTHRESH	Load threshold.
QTHRESH	Reaction force threshold.

REDUCE - *Phase 2 Reduction to Retained Degrees of Freedom*

Purpose

This command performs a Guyan matrix reduction process for a specified component substructure, otherwise known as matrix condensation. It produces the same result as obtained by the specification of OMIT or ASET data. The purpose is to reduce the size of the matrices. In static analysis only points on the boundary need be retained. In dynamics, the boundary points and selected interior points are retained.

Request Format

```
REDUCE name
```

Subcommands

```
NAME       new name (required)
BOUNDARY   b (required)
OUTPUT     m1, m2,...
RSAVE      (See Remark 4)
```

Definitions

Item	Description
name	Name of substructure to be reduced.
new name	Name of resulting substructure.
b	Set identification number of BDYC bulk data entries which define sets of retained degrees of freedom for the resulting reduced substructure matrices. See Remarks 1 and 2.
m1, m2, etc.	Optional output requests. See Remark 3.

Remarks

1. All references to the grid points and components not defined in the "boundary set" will be reduced out of the new substructure. Any subsequent reference to these omitted degrees of freedom in COMBINE, REDUCE, or SOLVE operations generates an error condition.
2. The same transformations will be applied to the reduced mass matrix for the new substructure. See the NASTRAN Theoretical Manual for a discussion of this effect.
3. The following output requests are available for the REDUCE operation:

Code	Output
1*	Current problem summary.
2	Boundary set summary.
3	Summary of Grid point ID numbers for each boundary set.
4	The EQSS item for the structure being reduced.
5*	The EQSS item.
6*	The BGSS item.
7	The CSTM item.
8	The PLTS item.
9*	The LODS item.

*.Recommended output options.

Requests 5-9 write formatted SOF items for the new reduced pseudo structure.

4. If the RSAVE directive is included, the decomposition product of the interior point stiffness matrix (LMTX item) is saved on the SOF file. This matrix will be used in the data recovery for the omitted points. If it is not saved, it will be regenerated when needed.

RESTORE - *Reload SOF*

Purpose

To reload the SOF from an external file created with the DUMP command.

Request Format

$$\text{RESTORE filename} \left[\begin{bmatrix}\text{TAPE}\\ \text{DISK}\end{bmatrix}\right]$$

Subcommands

None.

Definitions

Item	Description
filename	Name of the external file. Any one of the following: INPT, INP1,..., INP9.
TAPE	File resides on tape.
DISK	File resides on a direct access device.

Remarks

1. The external file must have been created with the DUMP command.
2. The SOF must be declared as NEW on the SOF command.
3. RESTORE must be the very first substructure command following the SOF and PASSWORD declarations.
4. The SOF size declarations for the RESTORE command must be exactly the same as for the SOF which was DUMPed. The DUMP/RESTORE commands cannot be used to increase the size of the SOF.

RUN - *Specifies Run Options*

Purpose

This command is used to limit the substructure execution for the purpose of checking the validity of the input data. It allows for the processing of input data separately from the actual execution of the matrix operations.

Request Format

$$\text{RUN}\left[\left[\begin{array}{c}\text{STEP}\\\text{DRY}\\\text{GO}\\\text{DRYGO}\end{array}\right]\right]$$

Subcommands

None.

Definitions

Item	Description
STEP	Will cause the execution of both DRY and GO operations one step at a time.
DRY	Limits the execution to table and transformation matrix generation. Matrix operations are skipped.
GO	Limits the execution to matrix generation only. This mode must have been preceded by a successful RUN=DRY or RUN=STEP execution.
DRYGO	Will cause execution of a complete dry run for the entire job, followed by a RUN=GO execution if no fatal errors were detected.

Remarks

1. The DRY, GO, and STEP options may be changed at any step in the input substructure command sequence. If the DRYGO option is used, the RUN directive must appear only once at the beginning.
2. If a fatal error occurs during the first pass of the DRYGO option, the program exits at the completion of all DRY operations.
3. The RUN = DRY option is handled differently for MREDUCE and CREDUCE because the matrix operations must be performed in order to generate the table and transformation matrix data. Input data only will be checked and no subsequent commands will be executed.
4. The RUN = GO and OPTIONS = K combination is illegal for any of the reduce operations, REDUCE, MREDUCE, or CREDUCE.

SOF - *Assigns Physical Files for Storage of the SOF*

Purpose

This declaration defines the names and sizes of the physical NASTRAN-CORE files you assign for storage of the SOF file. At least one of these declarations must be present in each substructure command deck. As many SOF declarations are required in the substructure command deck on each run as there are physical files assigned for the storage of the SOF file.

Request Format

$$\text{SOF(no.)} \ = \ \text{filename, filesize,} \left[\begin{bmatrix} \text{OLD} \\ \text{NEW} \end{bmatrix} \right]$$

Definitions

Item	Description
no.	Integer index of SOF file (1, 2, etc.) in ascending order of files required for storage of the SOF. The maximum index is 10. See Remarks 1, 2, and 3.
filename	User name for an SOF physical file. See Remarks 2, 3, and 7.
filesize	Size of allocated file space in kilowords, default = 100. See Remarks 1 and 4.
OLD	SOF data is assumed to already exist on the file.
NEW	The SOF is new. In this case, the SOF will be initialized. See Remark 5.

Remarks

1. If more space is required for storage of the SOF file, additional physical files may be declared. Alternatively, the file size parameter on a previously declared file may be increased, but only on the last physical file if more than one is used (on IBM the size of an existing file may not be increased).
2. Once an SOF declaration is made, the index of the SOF file must always be associated with the same file name. File names may not be changed from run to run.
3. The file name of each physical SOF file must be unique.
4. The declared size of the SOF may be reduced by the amount of contiguous free space at the end of the logical SOF file. This may be accomplished by removing the physical file declaration for those unused files which have the highest sequence numbers. An attempt to eliminate a portion of the SOF which contains valid data will result in a fatal error.
5. If the NEW parameter is present on any one of the SOF declarations, the entire logical SOF is considered new. Therefore, if an additional physical file is added to an existing SOF, the NEW parameter should not be included on any declarations.
6. You should insure that the correct SOF file is assigned for the current run. See the PASSWORD description.
7. The following conventions should be used for the file name declarations on each of the NASTRAN-COREcomputers:

   The filename used on the SOF declaration must specify one of the NASTRAN-CORE user files INPT, INP1,..., INP9. A maximum of 10 SOF file names is allowed in any NASTRAN-CORE substructuring run.

SOFIN - *Copy Items from File to SOF*

Purpose

To copy substructure items from an external file to the SOF.

Request Format

$$\text{SOFIN} \left[\begin{bmatrix}\text{INTERNAL}\\\text{EXTERNAL}\end{bmatrix}\right] \text{filename} \left[\begin{bmatrix}\text{TAPE}\\\text{DISK}\end{bmatrix}\right]$$

Subcommands

POSITION	NOREWIND
	REWIND
NAMES	WHOLESOF
	SUBSTRUCTURE name
ITEMS	ALL
	MATRICES
	PHASE 3
	TABLES
	item name

Definitions

Item	Description
EXTERNAL	File was written on a different computer type.
INTERNAL	File was written with GINO on the same computer type.
filename	Name of the external file. If the file is in INTERNAL format, filename must specify INPT, INP1,...,INP9. If the file is in EXTERNAL format, filename must specify a FORTRAN unit by using the form FORT1, FORT2,...,FORT32.
DISK	File is located on a direct access device.
TAPE	File is located on a sequential access device.
POSITION	Specifies initial file position. REWIND — File is rewound. NOREWIND — Input begins at the current position.
NAMES	Identifies a substructure for which data will be read. If NAMES=WHOLESOF is coded, and no other NAMES subcommands appear for the current SOFIN command, all substructure items found on the external file from the point specified by the POSITION subcommand to the end-of-file are copied to the SOF.

Item	Description	
ITEMS	Identifies the data items which are to be copied to the SOF for each substructure specified by the NAMES subcommands.	
	ALL	All items.
	MATRICES	All matrix items.
	PHASE3	The UVEC, QVEC, and SOLN items.
	TABLES	All table items item name: name of an individual item.
	item name	Name of an individual item.

Remarks

1. Filename is required. The other SOFIN operands are optional.
2. All subcommands are optional.
3. The NAMES subcommand may appear up to five times for each SOFIN command.
4. If a substructure name of an item which is to be copied to the SOF does not exist on the SOF, it is added to the SOF. MDI pointers for higher level, combined substructures, and lower level substructures arc restored.
5. For the EXTERNAL form of this command all the items on the file are read in and added to the SOF. The POSITION subcommand should be specified as REWIND and user specifications for all other subcommands are ignored.
6. SOFOUT is the companion substructure command.
7. When an internal-formatted file is located on tape and extends over multiple reels, care should be taken when using the SOFIN command. The commands should be ordered so that all the desired data is retrieved on a single pass through the tape. The following suggestions are helpful:
    - Order the SOFIN command to obtain data in the order they exist on the tape. If this order is not known, the CHECK command will list the contents of the tape.
    - The first SOFIN command should specify POSITION = REWIND and all subsequent commands should use POSITION = NOREWIND.
    - The individual items should be requested by name. The ALL, MATRICES, TABLES, or PHASE3 specification should not be used for the ITEMS subcommand unless all the appropriate items are on the tape. If some are not present, the tape will be searched to the end of the last reel and subsequent commands will not be executable because they will attempt to rewind back to the first tape.
8. Only one item may appear as an ITEMS subcommand per NAMES subcommand. Selective items may be referenced by repeating the NAMES subcommand.

SOFOUT - *Copy Items from SOF to File*

Purpose

To copy substructure items from the SOF to an external file.

Request Format

$$\text{SOFOUT}\left(\begin{bmatrix}\text{INTERNAL}\\ \text{EXTERNAL}\end{bmatrix}\right)\text{ filename }\left[\begin{bmatrix}\text{TAPE}\\ \text{DISK}\end{bmatrix}\right]$$

Subcommands

POSITION	NOREWIND
	REWIND
NAMES	WHOLESOF
	SUBSTRUCTURE name
ITEMS	ALL
	MATRICES
	PHASE 3
	TABLES
	item name

Definitions

Item	Description	
EXTERNAL	File will be written so that it can be read on a different computer type.	
INTERNAL	File will be written with GINO.(default)	
filename	Name of the external file. If the file is in INTERNAL format, filename must specify INPT, INP1,...,INP9. If the file is in EXTERNAL format, filename must specify a FORTRAN unit by using the form FORT1, FORT2,...,FORT32.	
DISK	File is located on a direct access device.	
TAPE	File is located on a sequential access device.	
POSITION	Specifies initial file position. (See remark 6).	
	REWIND	File is rewound.
	NOREWIND	Output begins at the current position.
	EOF	File is positioned to the point immediately preceding the end-of-file mark.
NAMES	NAMES Identifies a substructure for which data will be read. If NAMES=WHOLESOF is coded, and no other NAMES subcommands appear for the current SOFOUT command, all substructure items found on the SOF are copied to the external file.	
ITEMS	Identifies the data items which are to be copied to the external file for each substructure specified by the NAMES subcommands.	
	ALL	All items.
	MATRICES	All matrix items.
	PHASE3	The UVEC, QVEC, and SOLN items.
	TABLES	All table items item name: name of an individual item.
	item name	Name of individual item.

Remarks

1. Filename is required. The other SOFOUT operands are optional.
2. All subcommands are optional.
3. The NAMES subcommand may appear up to five times for each SOFOUT command.
4. PLTS items of pseudo structures reference the PLTS items of the component basic substructures. Therefore, in order to save all data necessary to plot a pseudo structure, the PLTS items of its component basic substructures must be saved as well as the PLTS item of the pseudo structure.
5. For the external form of this command, POSITION = NOREWIND has the effect of positioning the file to the end-of-file.
6. POSITION = REWIND should be coded for the first write to a new file.
7. SOFIN is the companion substructure command.
8. Only one item may appear as an ITEMS subcommand per NAMES subcommand. Selective items may be referenced by repeating the NAMES subcommand.

Example:

```
SOFOUT (EXTERNAL) FORT25 DISK
POSITION = REWIND
NAMES = PART1
ITEMS = ALL
```

The above command will export all items for PART1L to FORTRAN unit 25 using a machine-independent format.

SOFPRINT - *Requests SOF File Verification*

Purpose

To print selected contents of the SOF file for data checking purposes.

Request Format

```
SOFPRINT (opt) name, item1, item2, etc.
```

Subcommands

None.

Definitions

Item	Description
opt	Integer, control option, default = 0.
	opt = 1: prints data items only.
	opt = 0: prints table of contents.
	opt = -1: prints both.
name	Name of substructure for which data is to be printed.
item1, item2	SOF item name, used only when opt ≠ 0, limit = 5. (See **Table 2-2** for the list of item names.)

Remarks

1. If only the table of contents is desired (opt = 0) this command may be coded:

   ```
   SOFPRINT TOC
   ```

   On the page heading for the table of contents, the labels are defined as follows:

Label	Description	
SS	Secondary substructure number (successor).	
PS	Primary substructure number (predecessor).	
LL	Lower level substructure number.	
CS	Combined substructure number.	
HL	Higher level substructure number.	
TYPE	Substructure type:	
	B	Basic substructure.
	C	Combined substructure.
	R	Guyan reduced substructure.
	M	Real modal reduced substructure.
	CM	Complex modal reduced substructure.

   Any of the above types will have a prefix "I" if it is an image substructure resulting from an EQUIV operation.

SOLVE - *Substructure Solution*

Purpose

This command initiates the substructure solution phase. The tables and matrices for the pseudo structure are converted to their equivalent NASTRAN-CORE data blocks. The substructure grid points referenced on bulk data entries SPCS. MPCS, etc., are converted to pseudo structure scalar point identification numbers. The NASTRAN-CORE execution then proceeds as though a normal structure were being processed.

Request Format

```
SOLVE name
```

Subcommands

None. (Case Control and bulk data decks control the operations.)

Definitions

Item	Description
name	Name of pseudo structure to be analyzed.

Remarks

1. The allowable NASTRAN-CORE solutions are 1, 2, 3, 8, and 9.
2. Before requesting a SOLVE, you should check to be sure that all necessary matrices are available on the SOF file. For instance, loads and stiffness matrices are necessary in statics analysis. Mass and stiffness matrices are necessary in eigenvalue analysis, etc.
3. If the OPTIONS command has been used, an additional OPTIONS command may be necessary to ensure that the matrices required are available for the SOLVE operation.
4. Static load combinations of the original Phase 1 load vectors may be defined by the bulk data entry LOADC. Loads of this type may be used in Rigid Format 9 (Direct Transient Analysis) in lieu of DAREA dynamic load data.
5. The SOLVE name command should always be followed by RECOVER name to assure the solution data are saved on the SOF.
6. The SOLVE command may only be used in Phase 2 executions.

SUBSTRUCTURE - *Initiates the Substructure Control Data Deck*

Purpose

This command initiates the processing for automated substructuring and defines the phase of the analysis. It must be the first entry in the Substructure Control Deck.

Request Format

$$\text{SUBSTRUCTURE} \left[\begin{bmatrix} \text{PHASE1} \\ \text{PHASE2} \\ \text{PHASE3} \end{bmatrix} \right]$$

Subcommands

```
NAME      name (required and valid only in PHASE1)
SAVEPLOT  n (used only in PHASE1)
```

Definitions

Item	Description
name	The name assigned to the basic substructure which is being created in PHASE1.
n	The plot set identification used to define the set of elements and grid points to be saved in PHASE1 for subsequent plotting in PHASE2. Only one set may be defined for any basic substructure.

Remarks

1. The mode command RUN = STEP is assumed initially if the explicit command is not given immediately following the SUBSTRUCTURE command.
2. No further substructure commands are required for PHASE1.
3. Additional substructure commands are required for PHASE2.
4. For PHASE3 operations, RECOVER and BRECOVER are equivalent commands and one of them must be present.
5. Imbedded blanks within the individual elements of this directive are not allowed. An unrecognizable command causes the program to automatically assume a PHASE2 solution.

Automatic Substructure DMAP Alters

In the automated substructure process, commands are converted to the form of DMAP statements via ALTER statement equivalents. This section describes the resulting DMAP data for each command.

The raw DMAP data, stored in the program and modified according to user input, is listed by command type. The subcommand controls are identified by parentheses on the right side. For example, the (P only) for the SUBSTRUCTURE command item 12, implies that this DMAP statement is included only if the OPTION request includes P (loads).

The ALTER images are not true DMAP statements but are used to locate positions in the existing DMAP Solution Sequence for replacement by or insertion of the new DMAP statements. The relevant section of the Solution Sequence for each ALTER is indicated by the note in parentheses. For instance, "After GP4" in Solution Sequence 1 (statics) implies "ALTER nn" (where nn is the DMAP statement number of the GP4 module) for insertion of the corresponding DMAP statements following Solution Sequence 1 DMAP statement number nn. If an existing set of DMAP statements is to be removed, the parenthetical note may indicate "Remove DECOMP", where DECOMP may be a set of NASTRAN-CORE modules related to the entire decomposition process.

The descriptions given below are highly dependent on user input commands and the Solution Sequence selected. For an exact listing of all DMAP data generated for the current set of substructure commands, the DIAG 23 Executive Control may be input. Adding DIAG 24 will write the actual ALTERs generated on the system punch file. This feature allows these ALTERs to be modified and executed under APP DMAP,SUBS.

Index of Substructure DMAP ALTERs

BRECOVER - *(Phase 3)*

The BRECOVER command converts the results of a Phase 2 substructure analysis to NASTRAN-CORE solution vectors for the detailed calculation of basic structure (or an equivalent basic substructure) displacements, forces, loads, and stresses. The same structure model of the primary substructure defined in Phase 1 must be used in Phase 3. It is possible to perform the Phase 3 execution either as a restart of the Phase 1 run or as an independent run, which recalculates the necessary data blocks.

• *DMAP*

```
ALTER (Remove solution)
PARAM //*NOP*/ALWAYS=-1 $
SSG1 SLT,BGPDT,CSTM,SIL,EST,MPT,GPTT,EDT,MGG,CASECC,DIT/
PG/LUSET/NSKIP $
 (R.F. 9 only) Š (P or PA
SSG2 USET,GM,YS,KFS,GO,,PG/ Š only) QR,PO,PS,PL $
 (R.F. 1,2,3 or 9 only)
RCOVR3 ,PG,PS,PO,YS/UAS,QAS,PGS,PSS,POS,YSS,LAMA/SOLN/
*NAME*/NDUE $
EQUIV PGS,PG/ALWAYS $
EQUIV PSS,PS/ALWAYS $
EQUIV POS,PO/ALWAYS $
EQUIV YSS,YS/ALWAYS $
 (R.F. 1 or 2 only)
 (P or PA only)
COND LBSSTP,OMIT $
FBS LOO,,POS/UOOV/1/1/PREC/0 $
LABEL LBSSTP $
OFP LAMA,,,,,//CARDNO $
 (R.F. 3 only)
ALTER (After SDRI)
UMERGE USET,QAS,/QGS/*G*/*A*/*O* $
ADD QG,QGS/QGT $
EQUIV QGT,QG/ALWAYS $
EQUIV CASECC,CASEXX/ALWAYS $
ø (R.F. 8 or 9 only) ALTER (Remove repeat logic) Ÿ
```

• *Variables*

Name	Description
YS, PO	Remove if not P or PA, or if not Solution Sequence 1 or 2.
PG, PS	Remove if not P or PA, or if not Solution Sequence 1, 2, or 9.
UAS	ULV for Solution Sequence 1 and 2
	UDVF for Solution Sequence 8, UDVT for Solution Sequence 9
PGS	PGS for Solution Sequence 1 and 2, PPT for Solution Sequence 9
PSS	PSS for Solution Sequence 1 and 2, PST for Solution Sequence 9
LAMA	LAMA for Solution Sequence 3,
	PPF for Solution Sequence 8,
	TOL for Solution Sequence 9

Name	Description
QG	QG for Solution Sequence 1, 2, and 3, QPC for Solution Sequence 8 QP for Solution Sequence 9
POS	Remove if not P or PA. or if not Solution Sequence 1, 2, or 3.
SOLN	Solution Sequence solution number.
NAME	Name of basic Phase 1 substructure, corresponding to input data.
NOUE	Remove if not Solution Sequence 8 or 9.
STP	Step number.
PREC	Precision.

COMBINE - *Phase 2*

The COMBINE command initiates the process for combining several substructures defined on the SOF files. The COMB1 module reads the control deck and the bulk data and builds the tables and transformation matrices for the combination structure. The COMB2 module performs the matrix transformations using the matrices stored on the SOF file or currently defined as NASTRAN-CORE data blocks. The resultant matrices are stored on the SOF file and retained as NASTRAN-CORE data blocks.

• *DMAP*

```
COMB1 CASECC,GEOM4//STP/S,N,DRY/*PVEC* $
COND LBSTP,DRY $
COMB2 ,KN0l,KN02,KN03,KN04,KN05,KN06,KN07/KNSC/S,N,DRY ⌀
/*K*/* */*NAME0001*/*NAME0002*/*NAME0003*/ (K only)
*NAME0004*/*NAME0005*/*NAME0006*/*NAME0007* $
SOFO ,KNSC,,,,//S,N,DRY/*NAMEC */*KMTX* $
COMB2 ,MN01,MN02,MN03,MN04,MN05,MN06,MN07/MNSC/S,N,DRY/ ⌀
*M*/* */*NAME000l*/*NAME0002*/*NAME0003*/ (M only)
*NAME0004*/*NAME0005*/*NAME0006*/*NAME0007* $
SOFO ,MNSC,,,,//S,N,DRY/*NAMEC */*MMTX* $
COMB2 ,PN01,PN02,PN03,PN04,PN05,PN06,PN07/PNSC/S,N.DRY/ ⌀
*P*/*PVEC*/*NAME0001*/*NAME0002*/*NAME0003*/ (P or PA only
*NAME0004*/*NAME0005*/*NAME0006*/*NAME0007* $
SOFO ,PNSC,,,,//S,N,DRY/*NAMEC */*PVEC $
COMB2 ,BN0l,BN02,BN03,BN04,BN05,BN06,BN07/BNSC/S,N,DRY/ ⌀
*B*/* */*NAME0001*/*NAME0002*/*NAME0003*/ (B only)
*NAME0004*/*NAME0005*/*NAME0006*/*NAME0007* $
SOFO ,BNSC,,,,//S,N,DRY/*NAMEC */*BMTX* $
COMB2 ,K4N01,K4N02,K4N03,K4N04,K4N05,K4N06,K4N07/K4NSC/ ⌀
S,N,DRY/*K4*/* */*NAME000l*/*NAME0002*/*NAME0003*/ (K4 only)
*NAME0004*/*NAME0005*/*NAME0006*/*NAME0007* $
SOFO ,K4NSC,,,,//S,N,DRY/*NAMEC */*K4MX* $
LABEL LBSTP $
LODAPP PNSC,//*NAMEC */S,N,DRY $ (PA only)
```

• *Variables*

Name	**Description**
STP	Step number.
PVEC	PVEC for P option, PAPP for PA option.
N01,N02,...etc.	Internal numbers for structures to be combined.
NSC	Internal number of combined structure.
NAME0	Name of combined structure.

CREDUCE -

The CREDUCE command performs a complex modal synthesis reduction for a component substructure. The resulting generalized coordinates for the reduced substructure will consist of selected boundary point displacements and generalized displacements of the eigenvectors. The MRED1 module produces dummy USET and EED data blocks for the execution of the eigenvector extraction procedure. The EQST data block is created for use by the CMRED2 module. The CMRED2 module performs the actual matrix reduction. Note that, because the number of modal degrees of freedom is a calculated value, the RUN = DRY option is not allowed for complex modal reduction.

• *DMAP*

```
1 PARAM //*NOP*/ALWAYS=-1 $
2 MRED1 CASECC,GEOM4,DYNAMICS,CSTM/USETR,EEDR,EQST,DMR/*NAMEA */ 3
 S,N,DRY/STP/S,N,NOFIX/S,N,SKIPM/*COMPLEX* $
4 COND LBM3STP,DRY $
5 SOFI /KNOA,MNOA,PNOA,BNOA,K4NOA/S,N,DRY/*NAMEA */*KMTX*/*MNTX*/ 6
 *PVEC*/*BMTX*/*K4MX* $
7 COND LBM2STP,SKIPM $
ø 8 EQUIV KNOA,KFFX/NOFIX $
(K only) Š 9 EQUIV MNOA,MFFX/NOFIX $
(M only) Š 10 EQUIV BNOA,BFFX/NOFIX $
(B only) Š 11 EQUIV K4NOA,K4FFX/NOFIX $
(K4 only) Š 12 COND LBM1STP,NOF1X $
Š 13 SCE1 USETR,KNOA,MNOA,BNOA,K4NOA/KFFX,KFSX,KSSX,MFFX, Š 14
 BFFX,K4FFX $
Š (Remove for 15 LABEL LBM1STP $
Š option PA) 16 PARAMR //*COMPLEX*//1,0/GPARAM /G $
Š 17 ADD KFFX,K4FFX/KDD/G/(0,0,1,0) $
Š 18 EQUIV KDD,KFFX/ALWAYS $
Š 19 CEAD KFFX,BFFX,MFFX,EEDR,/PHIDR,CLAMA,OCEIGS,PHIDL Š 20 /NEIGVS
 $
Š 21 OFP CLAMA,OCEIGS,,,,// $
Š 22 EQUIV PHIDR,PHIFR/NOFIX $
Š 23 EQUIV PHIDL,PHIFL/NOFIX $
Š 24 COND LBM2STP,NOFIX $
Š 25 UMERGE USETR,PHIDR,/PHIFR/*N*/*F*/*S* $
Š 26 UMERGE USETR,PHIDL,/PHIFL/*N*/*F*/*S* $
Ÿ 27 LABEL LBM2STP $
28 CMRED2
 CASECC,CLAMA,PHIFR,PHIFL,EQST,USETR,KNOA,MNOA,BNOA,K4NOA,PNOA/ 29
 KNOB,MNOB,BNOB,K4NOB,PNOB,PONOB/STP/S,N,DRY/*PVEC* $
30 LABEL LBM3STP $
31 LODAPP PNOB,PONOB//*NAMEB___*/S,N,DRY $
(PA only) 32 COND FINIS,DRY $
```

• *Variables*

Name	Description
STP	Step number.
PVEC	PVEC for option P, PAPP for option PA.
NAMEA	Name of input substructure, A.
NAMEB	Name of output substructure, B.

Name	Description
NOA	Internal number of substructure A.
NOB	Internal number of substructure B.
KFFX, KFSX, KSSX	K only.
MFFX	M only.
BFFX	B only.
K4FFX	K4 only.
CLAMA, PHIFR, PHIFL	Remove for option PA.

SOFUT - *SOF Utility Command*

Several internal operations of the SOF may be performed with the utility commands which create various calls to the SOFUT module. Each of the commands and associated data are inserted as parameters.

• *DMAP*

```
SOFUT //DRY/*NAME */*OPER*/OPT/*NAME0002*/*PREF*/*ITM1*/*ITM2*/
2*ITM3*/*ITM4*/*ITM5* $
```

• *Variables*

Name	Description
NAME	Name of substructure.
OPER	Operation to be performed (first four characters of command, for example, EDIT).
OPT	Integer option code.
NAME0002	Second substructure name for EQUIV and RENAME.
PREF	Prefix for EQUIV operation.
ITM1, ITM2, etc.	SOF data item names.

The following table describes the variables used for each command.

Command Name	Oper Opt	NAME0002	PREF	ITM1, etc.
DELETE	X	X	X	
DESTROY	X	X		
EDIT	X	X	X	
EQUIV	X	X	X	X
RENAME	X	X	X	
SOFPRINT	X	X	X	X

MREDUCE

The MREDUCE command performs a modal synthesis reduction for a component substructure. The resulting generalized coordinates for the reduced substructure will consist of selected boundary point displacements and generalized displacements of the modal coordinates. The MRED1 module produces dummy USET and EED data blocks for the execution of the mode extraction procedure. The EQST and DMR data blocks are created for use by the MRED2 module. The MRED2 module performs the actual matrix reduction. Note that, because the number of modal degrees of freedom is a calculated value, the RUN = DRY option is not allowed for modal reduction.

• *DMAP*

```
MRED1 CASECC,GEOM4,DYNAMICS,CSTM/USETR,EEDR,EQST,DMR/*NAMEA */
S,N,DRY/STP/S,N,NOFIX/S,N,SKIPM/*REAL* $
COND LBM3STP,DRY $
SOFI /KNOA,MNOA,PNOA,BNOA,K4NOA/S,N,DRY/*NAMEA */*KMTX*/*MMTX*/
*PVEC*/*BMTX*/*K4MX* $
COND LBM2STP,SKIPM $
EQUIV KNOA,KFFX/NOFIX $ (K only)
EQUIV MNOA,MFFX/NOFIX $ (M only)
EQUIV BN0A,BFFX/NOFIX $ (B only)
EQUIV K4NOA,K4FFX/NOFIX $ (K4 only)
COND LBM1STP,NOFIX $
SCE1 USETR,KNOA,MNOA,BNOA,K4NOA/KFFX,KFSX,KSSX, (Remove for PA)
MFFX,BFFX,K4FFX $
LABEL LBM1STP $
READ KFFX,MFFX,BFFX,K4FFX,EEDR,USETR,/LAMAR,PHIR,
MIR,OEIGR/*MODES*/NEIGVS $
OFP LAMAR,OEIGR,,,,// $
EQUIV PHIR,PHIS/NOFIX $
COND LBM2STP,NOFIX $
UMERGE USETR,PHIR,/PHIS/*N*/*F*/*S* $
LABEL LBM2STP $
MRED2 CASECC,LAMAR,PHIS,EQST,USETR,KNOA,MNOA,BNOA,K4NOA,PNOA,DMR,
QSM/KNOB,MNOB,BNOB,K4NOB,PNOB,PONOB/STP/S,N,DRY/*PVEC* $
LABEL LBM3STP $
LODAPP PNOB,PONOB//*NAMEB */S,N,DRY $ (PA only)
COND FINIS,DRY $
```

• *Variables*

Name	Description
STP	Step number.
PVEC	PVEC for option P, PAPP for option PA.
NAMEA	Name of input substructure, A.
NAMEB	Name of output substructure, B.
NOA	Internal number of substructure A.
NOB	Internal number of substructure B.
KFFX, KFSX, KSSX	K only.
MFFX	M only.
BFFX	B only.
K4FFX	K4 only.

Name	Description
LAMAR, PHIS	Remove for option PA.
QSM	Remove for R.F. 9.

PLOT - *Substructure Plots:*

Any level of substructure may be plotted as an undeformed shape using the existing NASTRAN-CORE plot logic. The plot sets generated in Phase 1 are combined and transformed for that plotting.

• *DMAP*

```
PLTMRG CASECC,PCDB/PLTSTP,GPSTP,ELSTP,BGSTP,CASSTP,EQSTP/*NAME */
S,N,NGP/S,N,LSIL/S,N,NPSET $
SETVAL //S,N,PLTFLG/1/S,N,PFIL/0 $
PLOT PLTSTP,GPSTP,ELSTP,CASSTP,BGSTP,EQSTP,,,,,/PMSTP/NGP/LSIL/
S,N,NPSET/S,N,PLTFLG/S,N,PFIL $
PRTMSG PMSTP// $
```

• *Variables*

Name	Description
NAME	Name of substrcture to be plotted.
STP	Step number.

RECOVER (Phase 2) and MRECOVER (Phase 1, 2)

RECOVER performs the recovery and output of the Phase 2 solution data. MRECOVER performs the recovery and output subsequent to a Phase 1 or 2 MREDUCE or CREDUCE operation. The NASTRAN-CORE solution displacement vector (either displacement vectors or eigenvectors) is transformed and expanded to correspond to the degrees of freedom of the selected component substructures. Each pass through the DMAP loop corresponds to a requested structure to be processed. The RCOVR module selects the substructure to be processed with the loop counter, ILOOP.

• *DMAP*

```
FILE U1=APPEND/U2=APPEND/U3=APPEND/U4=APPEND/U5=APPEND $
PARAM //*ADD*/ILOOP/0/0 $
LABEL LBSTP $
RCOVR CASESS,GEOM4,KGG,MGG,PGG,UGV,DIT,DLT,BGG,K4GG,PPF/OUGV1,
OPG1,OQG1,U1,U2,U3,U4,U5/S,N,DRY/S,N,ILOOP/STP/*NAMEFSS */
NSOL/NEIGV/S,N,LUI/S,N,U1N/S,N,U2N/S,N,U3N/S,N,U4N/S,N,U5N/
S,N,NOSORT2/V,Y,UTHRESH/V,Y,PTHRESH/V,Y,QTHRESH $
EQUIV OUGV1 ,OUGV /NOSORT2/OQG1,OQG/NOSORT2 $
EQUIV OPG1,OPG/NOSORT2 $ (R.F. 1, 2, 8, or 9 only)
COND NST2STP,NOSORT2 $
SDR3 OUGV1 ,OPG1,OQG1,,,/OUGV ,OPG,OQG,,, $
LABEL NST2STP $
OFP OUGV ,OPG,OQG,,,//S.N,CARDNO $
COND LBBSTP,ILOOP $
REPT LBSTP,100 $
LABEL LBBSTP $
SOFO ,U1,U2,U3,U4,U5//-1/*xxxxxxxx* $
```

• *Variables*

Name	Description
KGG	K option only.
MGG	M option only.
BGG	B option only.
K4GG	K4 option only.
GEOM4	GEOM4 (Solution Sequences 1,2,8,9), LAMA (Solution Sequence 3)
PGG	PGG (Solution Sequences 1,2), Blank for Solution Sequence 3,
	PPF (Solution Sequence 8), PPT (Solution Sequence 9)
UGV	UGV (Solution Sequences 1, 2, 8, 9)
	PHIG (Solution Sequence 3)
PPF	(Blank for Solution Sequences 1, 2, 3)
	PPF (Solution Sequence 8), TOL (Solution Sequence 9)
OUGV1	OUGV1 (Solution Sequences 1, 2, 8, 9)
	OPHIG1 (Solution Sequence 3)
OUGV	OUGV (Solution Sequences 1, 2, 8, 9)
	OPHIG (Solution Sequence 3)
SS SS or CC	(if after SOLVE step).
DIT, DLT	Remove if not R.F. 1, 2, or 3.
OPG1, OPG	Remove if R.F. 3.
NSOL	Solution Sequence solution number.

Name	Description
NEIGV	R.F. 3 only.
NAMEFSS	Name of solution structure.

REDUCE

The REDUCE command initiates the matrix partitioning operations to be performed on the stiffness, mass, damping, and load vectors in order to produce a set of matrices defined by a subset of the original degrees of freedom. The REDUCE module generates the partitioning vector PV, a USET data block US, and an identity matrix IN from the bulk data and the corresponding substructure tables stored on the SOF. The remainder of the DMAP sequence directs the actual matrix operations.

• *DMAP*

```
REDUCE CASECC,GEOM4/PVNOA,USSTP,INSTP/STP/S,N,DRY/*PVEC* $
COND LBRSTP,DRY $
SOFI /KNOA,MNOA,PNOA,BNOA,K4NOA/S,N,DRY/*NAME000A*/*KMTX*/*MMTX*/
*PVEC*/*BMTX*/*K4MX* $
COND LBRSTP,DRY $
SMP1 USSTP,KNOA,,,/GONOA,KNOB,KONOA,LONOA,,,,, $
MERGE GONOA,INSTP,,,,PVNOA/GNOA/1/TYP/2 $ (K only)
SOFO ,GNOA,LONOA,,,//DRY/*NAME000A*/*HORG*/*LMTX* $
SOFO ,KNOB,,,,//DRY/*NAME000B*/*KMTX* $
SOF1 /GNOA,,,,/S,N,DRY/*NAME000A*/*HORG* $ (all except K)
MPY3 GNOA,MNOA,/MNOB/0/0 $ (M only)
SOFO ‘MNOB,,,,//DRY/*NAME000B*/*MMTX* $
MPY3 GNOA,BNOA,/BNOB/0/0 $ (B only)
SOFO ,BNOB,,,,//DRY/*NAME000B*/*BMTX* $
MPY3 GNOA,K4NOA,/K4NOB/0/0 $ (K4 only)
SOFO ,K4NOB,,,,//DRY/*NAME000B*/*K4MX* $
PARTN PNOA,,PVNOA/PONOA,,,/1/1/2 $ (P or PA only)
MPYAD GNOA,PNOA,/PNOB/1/1/0/1 $
SOFO ,PONOA,,,,//DRY/*NAME000A*/*POVE* $
SOFO ,PVNOA,,,,//DRY/*NAME000A*/*UPRT* $
S9F9 ,PNOB,,,,//DRY/*NAME000B*/*PVEC* $ (P or PA only)
LABEL LBRSTP $
LODAPP PNOB,PONOA//*NAME000B*/S,N,DRY $ (PA only)
```

• *Variables*

Name	Description
STP	Step number.
NAME000A	Name of input structure, A.
NAME000B	Name of output structure, B.
NOA, NOB	Internal numbers of substructures A and B.
TYP	Matrix precision flag (1 = single).
PVEC	PVEC for P option, PAPP for PA option.
POVE	POVE for P option, POAP for PA option.

RUN

The RUN command defines the DRY parameter for use by the subsequent DMAP statements. If you specify RUN = DRY, a special set of DMAP statements is placed at the end of the entire command sequence.

• *Raw DMAP*

```
PARAM //*ADD*/DRY/I /0$
```

• *Variables*

Name	Description

I	Integer code for RUN option (DRY = -1, GO = 0, STEP = 1).
	If RUN = DRYGO, I is set to (DRY) initially and the following DMAP is inserted at the end of the complete ALTER stream:
	`LABEL LBSEND $`
	`PARAM //*ADD*/DRY/DRY/1 $`
	`COND FINIS,DRY $`
	`REPT LBSBEG,1 $`
	`JUMP FINIS $`

EXIO - External I/O Commands

Several operations may be performed on the NASTRAN-CORE user files and the SOF file using the EXIO module. The various input parameters are set by the Substructure Commands.

• *DMAP*

```
EXIO //S,N,DRY/MACH/*DEVI*/*UNITNAME*/*FORM*/*MODE*/*POSI*/*ITEM*/
 *NAME0001*/*NAME0002*/*NAME0003*/*NAME0004*/*NAME0005* $
```

• *Variables*

Name	Description
MODE	First four characters of command name (that is, "SOFI", "REST").
DEVI	Device used for I/O file ("TAPE" or "DISK").
UNITNAME	Name of NASTRAN-CORE user file assigned to I/O file (that is, INPT, INP1, etc.).
FORM	Format of data ("EXTE" or "INTE").
POSI	Position of file on device ("REWI", "NORE", or "EOF").
ITEM	Name of SOF item or "ALL", "MATR", "TABL", or "PHAS". NAME0001, etc. Names of substructures to be copied.

The following table describes the variables used for each command:

Command	MODE	DEVI	UNITNAME	FORM	POSI	ITEM	NAME000i
SOFlN	X	X	X	X	X	X	X
SOFOUT	X	X	X	X	X	X	X
RESTORE	X	X	X				
DUMP	X	X	X				
CHECK	X	X	X				

SOLVE

The SOLVE command provides the necessary data for execution of the solution phase of NASTRAN-CORE. Module SGEN replaces the NASTRAN-CORE GP1 module for the purpose of defining an equivalent pseudo structure from data blocks. The new data blocks GE3S and GE4S contain the load and constraint data in the form of converted bulk data images. The stiffness, mass, viscous damping, and structural damping matrices are obtained from the SOF files and added to any user matrix terms. The static and dynamic analysis Solution Sequences require separate raw DMAP. Both sets of raw DMAP are shown below.

• *DMAP, Solution Sequences 1-3*

```
1 ALTER (Remove GP1) 2 PARAM //*NOP*/ALWAYS=-1 $
3 SGEN CASECC,GEOM3,GEOM4,DYNAMICS/CASESS,CASEI,GPL,EQUEXIN,GPDT, 4
  BGPDT,SIL,GE3S,GE4S,DYNS/S,N,DRY/*NAMESOLS*/S,N,LUSET/ 5 S,N,NOGPDT
  $
6 PURGE CSTM $
7 EQUIV GE3S,GEOM3/ALWAYS/GE4S,GEOM4/ALWAYS/CASEI,CASECC/ALWAYS/ 8
  DYNS,DYNAMICS/ALWAYS $
9 COND LBSTP,DRY $
10 ALTER (Remove PLOT) 11 ALTER (Remove NOSIMP COND) 12 COND
  LBSOL,NOSIMP $
13 ALTER (Remove Property Optimization EQUIV or NOMGG COND) 14 COND
  LBSOL,NOMGG $
15 ALTER (Remove SMA3) 16 LABEL LBSOL $
17 SOFI /KNOS,MNOS,,,/DRY/*NAMESOLS*/*KMTX*/*MMTX* $
18 EQUIV KNOS,KGG/NOSIMP $
(K only) 19 EQUIV MNOS,MGG/NOSIMP $
(M only) 20 COND LBSTP,NOSIMP $
21 ADD KGGX,KNOS/KGG $
(K only) 22 ADD MGG,MNOS/MGGX $
(M only) 23 EQUIV MGGX,MGG/ALWAYS $
24 LABEL LBSTP $
25 CHKPNT MGG $
26 ALTER (After GP4) 27 COND LBSEND,DRY $
28 ALTER (Remove SDR2 - PLOT)
```

• *Variables*

Name	Description
NAMESOLS	Name of solution structure.
NOS	Internal number of solution structure.
STP	Step number.

• *Raw DMAP, Solution Sequences 8, 9*

```
1 ALTER (Remove GP1)
2 PARAM //*NOP*/ALWAYS=-1 $
3 SGEN CASECC,GEOM3,GEOM4,DYNAMICS/CASESS,CASEI,GPL,EQEXIN,GPDT,
4 BGPDT,SIL,GE3S,GE4S,DYNS/S,N,DRY/*NAMESOLS*/S,N,LUSET/
5 S,N,NOGPDT $
6 PURGE CSTM $
7 EQUIV GE3S,GEOM3/ALWAYS/GE4S,GEOM4/ALWAYS/CASEI,CASECC/ALWAYS
```

```
DYNS,DYNAMICS/ALWAYS $
COND LBSTP,DRY $
ALTER (Remove PLOT)
ALTER (Remove NOSIMP PURGE and COND)
ALTER (Remove GPWG and SMA3)
SOFI /KNOS,MNOS,BNOS,K4NOS,/DRY/*NAMESOLS*/*KMTX*/*MMTX*/*BMTX*/
*K4MX* $
EQUIV KNOS,KGG/NOKGGX $
COND LB2K,NOKGGX $ (K only)
ADD KGGX,KNOS/KGG $
LABEL LB2K $
EQUIV MNOS,MGG/NOMGG $
COND LB2M,NOMGG $
ADD MGG,MNOS/MGGX $ (M only)
EQUIV MGGX,MGG/ALWAYS $
LABEL LB2M $
EQUIV BN0S,BGG/NOBGG $
COND LB2B,NOBGG $
ADD BGG,BNOS/BGGX $ (B only)
EQUIV BGGX,BGG/ALWAYS $
LABEL LB2B $
EQUIV K4NOS,K4GG/NOK4GG $
COND LB2K4,NOK4GG $
ADD K4GG,K4NOS/K4GGX $ (K4 only)
EQUIV K4GGX,K4GG/ALWAYS $
LABEL LB2K4 $
LBSTP $
CHKPNT MGG,BGG,K4GG $
ALTER (Remove MDEMA, KDEK2 PARAM)
PARAM //*AND*/MDEMA/NQUE/NOM2PP $
PARAM //*ADD*/KDEK2/1/0 $ (K only)
PARAM //*ADD*/NOMGG/1/0 $ (M only)
PARAM //*ADD*/NOBGG/1/0 $ (B only)
PARAM //*ADD*/NOK4GG/1/0 $ (K4 only)
ALTER (Remove NOSIMP, NOGPDT EQUIV)
EQUIV K2DD,KDD/KDEK2 $
EQUIV M2DD,MDD/NOMGG $
EQUIV B2DD,BDD/NOBGG $
ALTER (Remove SDR2 and PLOT)
EQUIV UPVF,UPVC/NOA $
COND LBL19,NOA $
SDR1 USETD,,UDVF,,,GOD,GMD,,,,/UPVC,,/1/DYNAMICS $
LABEL LBL19 $
CMKPNT UPVC $
EQUIV UPVC,UGV/NOUE $
COND LBUE,NOUE $
UPARTN USET,UPVC/UGV,UEV,,/*P*/*G*/*E* $
LABEL LBUE $
```

• *Variables*

Variable	Description
NAMESOLS	Name of solution structure.
NOS	Internal number of solution structure.
STP	Step number.
UDVF	UDVF for R.F. 8, UDVT for R.F. 9.

SUBSTRUCTURE

The SUBSTRUCTURE command is necessary to initiate the automatic DMAP process. In Phase 1, the SUBPH1 module is used to build the substructure tables on the SOF from the NASTRAN-CORE grid point tables and the SOFO module is used to copy the matrices onto the SOF. In Phase 2 and Phase 3, the initial value of the DRY parameter is set and the DMAP sequence is initiated.

• *DMAP*

PHASE 1

```
ALTER 2,0
PARAM //*NOP*/ALWAYS=-1 $
SGEN CASECC,,,/CASESS,CASEI,,,,,,,,/S,N,DRY/*XXXXXXXX*/S,N,LUSET/
 S,N,NOGPDT $
EQUIV CASEI,CASECC/ALWAYS $
ALTER (After GP4)
PARAM //*ADD*/DRY-1 /0 $
LABEL LBSBEG $
COND LBLIS,DRY $(R.F. 1, 2, 3, and 9 only)
SSG1 SLT,BGPDT,CSTM,SIL,EST,MPT,GPTT,EDT,MGG,CASECC,DIT/ ⌀ (R.F.
 PG/LUSET/NSKIP $
 9 & P
CHKPNT PG $ Ÿ or PA
ALTER (Remove DECOMP) only)
SSG2 USET,GM,,KFS,GO,,PG/QR,PO,PS,PL $ ⌀ (R.F.
CHKPNT PO,PS,PL $ Ÿ 9 & P
LABEL LBLIS $ (R.F. 1, 2, 3, and 9 only) or PA
ALTER (Remove solution) only)
SUBPH1 CASECC,EQEXIN,USET,BGPDT,CSTM,GPSETS,ELSETS//S,N,DRY/
*NAME */PLOTID /*PVEC* $
COND LBSEND,DRY $
EQUIV PG,PL/NOSET $ ⌀ R.F. 1,
COND LBL10,NOSET $ Š 2, or 3
SSG2 USET,GM,YS,KFS,GO,,PG/QR,PO,PS,PL $ Š & P or
CHKPNT PO,PS,PL $ Ÿ PA only)
LABEL LBL10 R
SOFO ,KAA,MAA,PL,BAA,K4AA//S,N,DRY/*NAME*/*KMTX*/*MMTX*/PVEC*/
*BMTX*/*K4MX* $
LODAPP PL,//*NAME */S,N,DRY $ (R.F. 1, 2, 3, or 9 and PA only)
EQUIV CASESS,CASECC/ALWAYS $
```

PHASE 2

```
ALTER 2,0
PARAM //*ADD*/DRY/I/0 $
LABEL LBSBEG $
```

PHASE 3

```
ALTER (Remove DECOMP or before dynamic solution)
PARAM //*ADD*/DRY/I/0 $
LABEL LBSBEG $
```

• *Variables*

Name	Description
I	Integer
RUN	option code (see RUN command).
NAME	Phase 1 substructure name.
PLOTID	Phase 1 Plot Set ID.
KAA, MAA, PL, BAA, K4AA	Data blocks dependent on OPTION.
PVEC	PVEC for option P, PAPP for option PA.

3 CASE CONTROL

Introduction

The Bulk Data section uses the idea of sets of things. Things like loads and comstraints, for instance. Case Control then provides the means by which we can chunk the analysis flow into a number of separate segments using a Subcase structure. The set idea then allows eack Subcase to use a differnt set of loads and or constraints, for example. Using this structure, if you will, we can solve several similar problems in a single run. Those items that can be selected in this manner are defined in the following section.

Data Selection

The Case Control commands that are used for selecting items from the bulk data section are listed below in functional groups. A detailed description of each command is given in **Case Control Format**. The first four characters of the mnemonic are sufficient if unique.

Loads

A member of load sets is selected by one of the following commands:

Command Name	Description
DEFORM	Selects member of the element deformation set defined by DEFoRM bulk data entries
DLOAD	Selects a member of the dynamic load set defined by RLOAD and TLOAD bulk data entries
DSCOEFFICIENT	Selects loading factor for normal modes with differential stiffness
LOAD	Selects member of the static structural load set.
NONLINEAR	Selects nonlinear loading condition for transient response
PLCOEFFICIENT	Selects loading increments for piece wise linear analysis

Constraints

The following case control commands are used for the selection of constraints:

Command Name	Description
AXISYMMETRIC	Selects boundary conditions for conical shell and axisymmetric solid elements.
MPC	Selects set of multipoint constraints for structural displacement.
SPC	Selects set of single-point constraints for structural displacements.

Input Matrices

The following case control commands are used for the selection of direct input matrices:

Control Name	Description
B2GG	Selects direct input structural g-set damping
K2GG	Selects direct input g-set structural stiffness.
M2GG	Selects direct input g-set mass matrices
P2G	Selects direct input g-set loads
B2PP	Selects direct input structural p-set damping.
K2PP	Selects direct input structural p-set stiffness.
M2PP	Selects direct input p-set mass matrices
TFL	Selects transfer functions

Conditions for Dynamics

The following case control commands specify the conditions for dynamic analyses:

Control Name	Description
CMETHOD	Selects the conditions for complex eigenvalue extraction.
CUTOFF	Sets cutoff frequency for modal solutions
FREQUENCY	Selects the frequencies to be used for frequency and random response calculations.
IC	Selects the initial conditions for direct transient response.
METHOD	Selects the conditions for real eigenvalue analysis.
MODESELECT	Selects a subset of the computed modes for inclusion in subsequent analysis
RANDOM	Selects the power spectral density functions to be used in random analysis.
SDAMPING	Selects table to be used for determination of modal damping.
TSTEP	Selects time steps to be used for integration in transient response problems.

Thermal Loads

The following case control commands are associated with the use of thermal fields:

Control Name	Description
TEMPERATURE(LOAD)	Selects thermal field to be used for determining equivalent static loads.
TEMPERATURE(MATERIAL)	Selects thermal field to be used for determining structural material properties.
TEMPERATURE	Selects thermal field for determining both equivalent static loads and material properties.

Output Selection

Printer output requests may be grouped in packets following OUTPUT statements or the individual requests may be placed anywhere in the Case Control section prior to any structure plotter or curve plotter requests. Plotter requests are described in **Chapters 5 and 6**. The Case Control statements that are used for output selection are listed below in functional groups.

Titling Directives

The following are associated with output control, titling and bulk data echoes:

Control Name	Description
ECHO (p. 138)	Selects echo options for Bulk Data section, default is a sorted bulk data echo. Note: Echoes of the Executive Control and the Case Control sections are automatically printed and cannot be suppressed.
LABEL (p. 156)	Defines a text string to be printed on third line of each page of output
LINE (p. 157)	Sets the number of data lines per printed page, default is 50
MAXLINES (p. 161)	Sets the maximum number of output lines, default is 20000
SUBTITLE (p. 200)	Defines a text string to be printed on second line of each page of output
TITLE (p. 208)	Defines a text string to be printed on first line of each page of output

Calculated Quantities

The following are used in connection with some of the specific output requests for calculated quantities:

Control Name	Description
MODESELECT (p. 166)	Selects a set of modes to be kept for subsequent analysis
OFREQUENCY (p. 173)	Selects a set of frequencies to be used for output requests in frequency response problems (default is all frequencies) or flutter velocities
OTIME (p. 175)	Selects a set of times to be used for output requests in transient analysis problems (default is all times)
SET (p. 191)	Defines lists of point numbers, elements numbers, or frequencies for use in output requests
TSTEP (p. 209)	Selects a set of time steps to be used for output requests in transient response problems

Unique Dynamic solution output

The following is used to make output requests for the calculated data in a modal dynamic solution:

Control Name	Description
EDE (p. 139)	Requests the calculation and printout of Element Energy Loss (Element Damping Energy) - currently only available in frequency response
EKE (p. 141)	Requests the calculation and printout of Element Kinetic Energy - currently only available in normal modes and frequency response
GPKE (p. 150)	Calculate Grid Point Kinetic Energy for normal modes
MEFFMASS (p. 162)	Requests the calculation and printout of modal effective mass, participation factors and additional information, which can be useful in identifying modes.
MODALSE (p. 164)	Requests the calculation and printout of modal contributions in a frequency response solution.

Response Variables for SOLUTION Set

The following are used to make output requests for the calculated response of components in the SOLUTION set (components in the direct or modal formulation of the general K system) for dynamics problems:

Control Name	Description
SACCELERATION	Requests the acceleration of the independent components for a selected set of points or modal coordinates.
SDISPLACEMENT	Requests the displacements of the independent components for a selected set of points or modal coordinates.
SVELOCITY	Requests the velocities of the independent components for a selected set of points or modal coordinates.

Forces and Stresses and Nodal Response

The following are used to make output requests for stresses and forces, as well as calculated response of degrees of freedom used in the model:

Control Name	Description
ACCELERATION	Requests the accelerations for a selected set of PHYSICAL points (grid and scalar plus extra points introduced for dynamic analysis)
DISPLACEMENT	Requests the displacements for a selected set of PHYSICAL points.
ESE	Requests structural element strain energies in static analysis, normal modes, and frequency response analysis.
FORCE or ELFORCE	Requests the forces in a set of structural elements.
GPFORCE	Requests grid point force balance due to element forces, forces of single point constraint, and applied loads in static analysis.
HARMONICS	Controls the number of harmonics that will be output for requests associated with the conical shel and axisymmetric solids
MPCFORCE	Requests multipoint forces of constraint at a set of points in APP DISP solution sequences 1, 2, 3, 14, and 15
NCHECK	Requests significant digits to indicate numerical accuracy of element stress and force computations.
NLLOAD	requests the nonlinear loads for a selected set of physical points (grid points and extra points introduced for dynamic analysis) in transient response problems.
OLOAD	Selects a set of applied loads for output
SCAN	SCANs output data and eliminates values that do not meet the specification set by this SCAN command
SPCFORCES	Requests the single-point forces of constraint at a set of points.
STRAIN	requests the strains/curvatures in a set of structural elements (applicable to TRIA3 and QUAD4 only)
STRESS or ELSTRESS	Requests the stresses in a set of structural elements.
VECTOR	Requests displacements for a selected set of PHYSICAL points
VELOCITY	Requests the velocities for a selected set of PHYSICAL points.

Subcase Definition

In general, a separate subcase is defined for each loading condition. In statics problems separate subcases are also defined for each set of constraints. In complex eigenvalue analysis and frequency response separate subcases are defined for each unique set of direct input matrices. Subcases may be used in connection with output requests, such as in requesting different output for each mode in a real eigenvalue problem.

The Case Control Section is structured so that a minimum amount of repetition is required. Only one level of subcase definition is necessary. All items placed above the subcase level (ahead of the first subcase) will be used for all following subcases, unless overridden within the individual subcase.

In statics problems, subcases may be combined through the use of the SUBCOM feature. Individual loads may be defined in separate subcases and then combined by the SUBCOM. If the loads are mechanical, the responses are combined as shown in example 2, which follows. If a thermal load is involved, the responses due to mechanical and thermal loads may be recovered as shown in example 1. By redefining the thermal load(s) at the SUBCOM level, stresses and forces may be recovered.

In statics problems, provision has been made for the combination of the results of several subcases. This is convenient for studying various combinations of individual loading conditions and for the superposition of solutions for symmetrical and anti symmetrical boundaries.

Typical examples of subcase definition are given following a brief description of the commands used in subcase definitions.

The following case control statements are associated with subcase definition:

Control Name	Description
MODES	Controls the output for a given subcase as specified by the number of modes, otherwise all modes will be used.
REPCASE	Defines a subcase in static analysis that is used to make additional output requests for the previous real subcase. This command is required because multiple output requests for the same item are not permitted in the same subcase. Output requests above the subcase level are still used.
SUBCASE	Defines the beginning of a subcase that is terminated by the next subcase delimiters encountered.
SUBCOM	Defines a combination of two or more immediately preceding subcases in statics problems. Output requests above the subcase level are used.
SUBSEQ	Defines the coefficients for making the linear combination of the preceding subcases and must appear in a subcase defined by SUBCOM.
SYM	Defines a subcase in statics problems for which only output requests within the subcase will be used. Primarily for use with symmetry problems where the individual parts of the solution may not be of interest.
SYMCOM	Defines a combination of two or more immediately preceding SYM subcases in static problems. Output requests above the subcase level are used.
SYMSEQ	May appear in a subcase defined by SYMCOM to give the coefficient for making the linear combination of the preceding SYM subcases. A default value of 1.0 is used if no SYMSEQ command appears.

The following examples of Case Control indicate typical ways of defining subcases:

1. Static analysis with multiple loads

```
OUTPUT
DISPLACEMENT = ALL
MPC = 3
SUBCASE 1
SPC = 2
TEMPERATURE(LOAD) = 101
LOAD = 11
SUBCASE 2
SPC = 2
DEFORM = 52
LOAD = 12
SUBCASE 3
SPC = 4
LOAD = 12
SUBCASE 4
MPC = 4
SPC = 4
```

Four subcases are defined in this example. The displacements at all grid points will be printed for all four subcases. MPC = 3 will be used for the first three subcases and will be overridden by MPC = 4 in the last subcase. Since the constraints are the same for subcases 1 and 2 and the subcases are contiguous, the static solutions will be performed simultaneously. In subcase 1, thermal load 101 and external load 11 are internally superimposed, as are the external and deformation loads in subcase 2. In subcase 4 the static loading will result entirely from enforced displacements of grid points.

2. Linear combination of subcases:

```
SPC = 2
OUTPUT
SET 1 = 1 THRU 10,20,30
DISPLACEMENT = ALL
STRESS = 1
SUBCASE 1
LOAD = 101
OLOAD = ALL
SUBCASE 2
LOAD = 201
OLOAD = ALL
SUBCOM 51
SUBSEQ = 1.0,1.0
SUBCOM 52
SUBSEQ = 2.5,1.5
```

Two static loading conditions are defined in subcases 1 and 2. SUBCOM 51 defines the sum of subcases 1 and 2. SUBCOM 52 defines a linear combination consisting of 2.5 times subcase 1 plus 1.5 times subcase 2. The displacements at all grid points and the stresses for the element numbers in SET will be printed for all four subcases. In addition, the nonzero components of the static load vectors will be printed for subcases 1 and 2.

3. Statics problem with one plane of symmetry.

```
OUTPUT
SET 1 = 1,11,21,31,51
SET 2 = 1 THRU 10, 101 THRU 110
DISPLACEMENT = 1
ELFORCE = 2   SYM 1
SPC = 11
LOAD = 21
OLOAD = ALL
SYM 2
SPC = 12
LOAD = 22
SYMCOM 3
SYMCOM 4
SYMSEQ 1.0,-1 .0
```

Two SYM subcases are defined in subcases 1 and 2. SYMCOM 3 defines the sum and SYMCOM 4 the difference of the two SYM subcases. The nonzero components of the static load will be printed for subcase 1 and no output is requested for subcase 2. The displacements for the grid point numbers in set 1 and the forces for elements in set 2 will be printed for subcases 3 and 4.

4. Use of REPCASE in statics problems.

```
SET 1 = 1 THRU 10, 101 THRU 110, 201 THRU 210
SET 2 = 21 THRU 30, 121 THRU 130, 221 THRU 230
SET 3 = 31 THRU 40, 131 THRU 140, 231 THRU 240
SUBCASE 1
LOAD =10
SPC = 11
DISPLACEMENT = ALL
SPCFORCE = 1
ELFORCE = 1
REPCASE 2
```

```
ELFORCE = 2
REPCASE 3
ELFORCE = 3
```

This example defines one subcase for solution and two subcases for output control. The displacements at all grid points and the nonzero components of the single-point forces of constraint along with forces for the elements in SET 1 will be printed for SUBCASE 1. The forces for elements in SET 2 will be printed for REPCASE 2 and the forces for elements in SET 3 will be printed for REPCASE 3.

5. Use of MODES in eigenvalue problems.

```
METHOD = 2
SPC = 10
SUBCASE 1
DISPLACEMENT = ALL
STRESS = ALL
MODES = 2
SUBCASE 3
DISPLACEMENT = ALL
```

In this example the displacements at all grid points will be printed for all modes. The stresses in all elements will be printed for the first two modes.

Case Control Format

Case Control commands can be abbreviated to the first four characters, provided that the obviation is unique. The format of the Case Control commands is free-field. In presenting general formats for each command embodying all options, the following conventions are used:

1. Describers in upper-case are keywords and must be input as shown.
2. Describers in lower-case letters indicate that an appropriate value must be provided.
3. Braces,{ } **,** indicate that a choice of contents is mandatory.
4. Brackets [ ] indicate that a choice is optional.
5. Describers that are underlined indicate defaults.
6. Each command line is limited to 72 characters. A line can be continued to the next line of input by ending the line with a comma.

Note: See the Acceleration Case Control command for annotations.

The structure plotter output request packet and the x-y output request packet, while part of the Case Control command, are treated separately in **Chapters 5 and 6**, respectively.

ACCELERATION - *Acceleration Output Request*

Description

Requests form and type of acceleration vector output.

Format and Examples

$$ACCELERATION\left[\left(\begin{bmatrix}SORT1\\SORT2\end{bmatrix},\begin{bmatrix}PRINT\\PUNCH\\PLOT\end{bmatrix},\begin{bmatrix}REAL\\IMAG\\PHASE\end{bmatrix}\right)\right]=\begin{Bmatrix}ALL\\n\\NONE\end{Bmatrix}$$

Note: The terms inside the brackets [] are optional. If keywords inside the [] are specified the parentheses are required.

```
ACCELERATION = 5
ACCELERATION(SORT2, PHASE) = ALL
ACCELERATION(SORT1, PRINT, PUNCH, PHASE) = 17
```

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point. SORT2 is available only in transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output for solutions having complex response.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) for solutions having complex response.
ALL	Accelerations for all points will be output.
n	Set identification of a SET command that must be defined prior in Case Control. Only accelerations of points whose identification numbers appear on this SET command will be output (Integer > 0).
NONE	Accelerations will not be output.

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative to this would be to define a SET of interest.
3. Acceleration output is only available for transient and frequency response problems.
4. In a frequency response problem any request for SORT2 output causes all output to be SORT2.
5. ACCELERATION = NONE allows overriding an overall output request.
6. SYSTEM cell 121 operates as a filter on the acceleration output.by changing any terms with a magnitude less than $1.x10^{SYSTEM(121)}$ to 0.0 or (0.0,0.0) for complex values

AXISYMMETRIC - *Boundary Conditions*

Description

Selects boundary conditions for problems containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format and Examples

$$AXISYMMETRIC = \begin{bmatrix} SINE \\ COSINE \\ FLUID \\ SIN1 \\ COS1 \end{bmatrix}$$

AXISYMMETRIC = COSINE

Option	Meaning
SINE	Sine boundary conditions will be used.
COSINE	Cosine boundary conditions will be used.
SYMM, ANTISYMM, ANOM, ANTIANOM, SYMMANON	Used in magneto statics problems.

Remarks

1. This command is required for problems containing the elements named above.
2. See **Section 1.3.6 of User's Manual** for a discussion of the conical shell problem.
3. See **Section 1.3.7 of User's Manual** for a discussion of the axisymmetric solid problem.
4. The sine boundary condition will constrain components 1, 3, and 5 at every ring for the zero harmonic.
5. The cosine boundary condition will constrain components 2, 4, and 6 at every ring for the zero harmonic.
6. SPC and MPC case control commands may also be used to apply additional constraints.
7. See PROLOATE bulk data entry for magnetostatic problem involving the prolate spheroidal surface harmonic expansion.

BEGIN BULK - *End of Case Control Section*

Description

A mandatory delimiter that defines the end of the Case Control section. Input appearing after this command are assumed to be Bulk Data entries.

Format and Examples

```
BEGIN BULK
```

B2GG - *Direct Input Damping Matrix Selection*

Description

Selects a direct input damping matrix.

Format and Examples

```
B2GG =  name
B2GG =  WINGDAMP
B2GG =  B2GG
```

Option	Meaning
name	The same character string assigned to a [B2GG] matrix on a DMIG or DMIAX bulk data entry.

Remarks

1. B2GG matrix input is added to the damping matrix associated with the g-set of displacements.
2. The B2GG matrix must be real and symmetric.
3. Terms of the B2GG matrix are added to the damping matrix generated by damping elements before constraints are applied.
4. DMIG and DMIAX matrices will not be used unless selected.
5. These matrices are added into the system matrices before processing begins and will be included in all processing. Effects of these matrices will show up in the Grid Point Weight Generator output and in the processing of AUTOSPC. These matrices cannot reference extra points (EPOINTs) and must be symmetric and real.

B2PP - *Direct Input Damping Matrix Selection*

Description

Selects a direct input damping matrix.

Format and Examples

```
B2PP  =  name
B2PP  =  BDMIG
B2PP  =  B2PP
```

Option	Meaning
name	The same character string assigned to a [B2pp] matrix on a DMIG or DMIAX bulk data entry.

Remarks

1. B2PP is used only in dynamics problems.
2. DMIG and DMIAX matrices will not be used unless selected.
3. These matrices are added to the system matrices after all reductions are performed and after the solution for the real (undamped) modes. Therefore, terms from these matrices will not be able to prevent mechanisms if a reduction is being done. Also, it is possible that AUTOSPC might constrain dof which these matrices connect to, as it is performed before these matrices are added to the system.These matrices may be symmetric or square and they may also be real or complex (only for frequency response and complex eigenvalue solutions). These matrices may reference any dof in the model (including EPOINTs).

CMETHOD - *Complex Eigenvalue Extraction Method Selection*

Description

Selects complex eigenvalue extraction data to be used for extraction of complex eigenvalues.

Format and Examples

```
CMETHOD = <SET>
CMETHOD = 101
```

Option	Meaning
n	Set identification of EIGC (and EIGP) bulk data entry (Integer > 0).

Remarks

1. Eigenvalue extraction data must be selected when extracting complex eigenvalues using functional module CEAD.

CUTOFF - *Set cutoff frequency for modal solutions*

Description

Sets cutoff frequency for modal solutions - allows truncation of modal dynamic response..

Format and Examples

$$CUTOFF\left\{\begin{matrix}\underline{MIN}\\MINMAX,\\SET\end{matrix}\frac{MASS}{NOMASS},\frac{DAMP}{NODAMP},\frac{STIFF}{NOSTIFF},\frac{K4}{NOK4}\right\} = \begin{matrix}\underline{FMIN}\\FMIN,FMAX\\SETID\end{matrix}$$

```
CUTOFF =  50.
CUTOFF(MINMAX)  =  25.3, 35.9
SET 999 = 1, 2, 4, 89.77
CUTOFF(SET,NOMASS,NOSTIFF)  = 999
```

Option	Meaning
MIN	All modes with a natural frequency above FMIN will have their terms in the selected matrices modified. see Remark 3
MINMAX	All modes with a natural frequency between FMIN and FMAX will have their terms in the dynamic matrices modified. See Remark 4
SET	All modes selected in the specified set (SETID) will have their terms in the dynamic matrices modified. See Remark 5
MASS	Remove any terms in the H-set mass matrix associated with the selected modes.
NOMASS	No modification to the H-set mass matrix.
DAMP	Remove any terms in the H-set viscous damping matrix associated with the selected modes.
NODAMP	No modification to the H-set viscous damping matrix.
STIFF	Remove any imaginary terms in the H-set stiffness matrix associated with the selected modes (see remark 1).
NOSTIFF	No modification to the H-set stiffness matrix.
K4	Remove any terms in the H-set structural damping matrix associated with the selected modes.
NOK4	No modification to the H-set structural damping matrix.
FMIN	Real - Cutoff frequency (Hz), see remarks 1,3
FMAX	REAL - Upper limit to cutoff frequency - matrix terms for modes with natural frequencies above FMAX will not be affected by this command.
SETID	Integer - set containing the modes which will be affected by this command

Remarks

1. This command is provided to allow user control over the dynamic response of high-frequency modes. By default all modes are allowed to respond dynamically to the input. If it is desired to truncate the dynamic response of high frequency modes, this command allows you to do that by modifying the dynamic matrices. The selected matrices are modified so that any modes selected will have no terms in the associated matrices (NOTE - if STIFF is selected, only the inaginary terms are removed - this allows for the static response of the modes using the real terms).
2. The H-set (solution set for modal solutions) are affected by this command. Only the selected matrices will be modified.

3. If MIN is chosen, the matrix terms for all modes with a natural frequency above FMIN (hz) will be affected by this command.
4. If MINMAX is chosen, the matrix terms for all modes with a natural frequency greater than FMIN, but less than FMAX will be affected by this command.
5. If SET is selected, the matrix terms for all modes listed in the selected set (SETID) will be affected by this command. NOTE: the set may contain both real and integer values. Integer values are the mode number in the solution set (This is after the application of the MODESELECT command and parameters LFREQ, HFREQ, and LMODES). Real values are the natural frequencies of the selected modes - the modes with natural frequencies closest to the values will be selected.
6. NOTE - in modal transient response, if this command is used to set modal mass of damping to 0.0 for selected modes, then the uncoupled (PARAM,NONCUP,0) solution must be used or the run will fail in TRD.
7. This command is stored in words 225 through 231 of CASECC.

DEFORM - *Element Deformation Static Load*

Description

Selects the element deformation set to be applied to the structural model.

Format and Examples

DEFORM = n

DEFORM = 27

Option	Meaning
n	Set identification of bulk data DEFORM entries (Integer > 0).

Remarks

1. DEFORM bulk data will not be used unless selected by the DEFORM Case Control command.
2. DEFORM is only applicable in statics, inertia relief, differential stiffness, and buckling problems.
3. The total load applied will be the sum of external, (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement loads (SPC).
4. Static, thermal, and element deformation loads should have unique identification numbers.

DISPLACEMENT - *Displacement Output Request*

Description

Requests form and type of displacement vector output.

Format and Examples

$$DISPLACEMENT\left[\left(\left\{\begin{matrix} SORT1 \\ SORT2 \end{matrix}, \frac{PRINT}{\frac{NOPRINT, PUNCH}{PLOT}}, \frac{REAL}{\frac{IMAG}{PHASE}}\right\}\right)\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

```
DISPLACEMENT  =  5
DISPLACEMENT(REAL)  =  ALL
DISPLACEMENT(SORT2, PUNCH, REAL)  =  ALL
```

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each grid point. SORT2 is available only in static analysis, transient, and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
NOPRINT	Displacement is calculated and saved on output file. The output file will not be sent to the output device.
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Displacements for all points will be output.
NONE	Displacements for no points will be output.
n	Set identification of previously appearing SET command. Only displacements of points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative to this would be to define a SET of interest.
3. In static analysis or frequency response problems, any request for SORT2 causes all output to be SORT2.
4. VECTOR is an alternate form and is entirely equivalent to DISPLACEMENT.
5. DISPLACEMENT = NONE allows overriding an overall output request.
6. SYSTEM cell 121 operates as a filter on the displacement output.by changing any terms with a magnitude less than $1.x10^{SYSTEM(121)}$ to 0.0 or (0.0,0.0) for complex values

DLOAD - *Dynamic Load Set Selection*

Description

Selects the dynamic load to be applied in a transient or frequency response problem.

Format and Examples

DLOAD = *n*

DLOAD = 73

Option	Meaning
n	Set identification of a DLOAD, RLOAD1, RLOAD2, TLOAD1, or TLOAD2 entries (Integer > 0).

Remarks

1. The above loads will not be used unless selected by the DLOAD Case Control command.
2. RLOAD1 and RLOAD2 may only be selected in a frequency response problem.
3. TLOAD1 and TLOAD2 may be selected in a transient or frequency response problem.

DSCOEFFICIENT - *Differential Stiffness Coefficient Set*

Description

Selects the coefficient set for a normal modes with differential stiffness problem.

Format and Examples

$$DSCOEFFICIENT = \begin{Bmatrix} DEFAULT \\ n \end{Bmatrix}$$

DSCOEF = 15
DSCOEF = DEFAULT

Option	Meaning
DEFAULT	A single default coefficient of value 1.0.
n	Set identification of DSFACT entry (Integer > 0).

Remarks

1. DSFACT bulk data entries will not be used unless selected by the DSCOEFFICIENT Case Control command.
2. DSCOEFFICIENT must appear in the second subcase of a normal modes with differential stiffness problem.

ECHO - *Bulk Data Echo Request*

Description

Requests echo of Bulk Data Deck.

Format and Examples

$ECHO = [SORT, UNSORT, BOTH, NONE, PUNCH]$

ECHO = BOTH

ECHO = PUNCH, SORT

Option	Meaning
SORT	Sorted echo will be printed.
UNSORT	Unsorted echo will be printed.
BOTH	Both sorted and unsorted echo will be printed.
NONE or NONO	No echo will be printed.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.

Remarks

1. If no ECHO command appears, ECHO = BOTH is assumed for restart runs. For all other runs, ECHO = SORT is assumed.
2. The request for sorted echo in a checkpoint run will be difficult to change in a subsequent restart run.
3. In a restart run, the unsorted echo lists only the new bulk data submitted with the run, while the sorted echo lists the resequenced and renumbered revised bulk data.
4. If CHKPNT YES is specified, a sorted echo will be printed unless ECHO = NONE.
5. Unrecognizable options will be treated as SORT.
6. Any option overrides the default. Thus, for example, if both print and punch are desired, both SORT and PUNCH must be requested on the same command.
7. The NONE option cannot be combined with the PUNCH option. If punch output only is desired, ECHO = PUNCH will suffice.
8. In a restart run, ECHO = NONO suppresses also the printing of the DMAP compiler source listing. Do not use ECHO = NONO and CHKPNT YES together.

EDE - *Element Energy Loss Output Request*

Description

Requests element energy loss per cycle output.

Format and Examples

$$EDE\left[\left(\frac{PRINT}{PUNCH}\right)\right]\left[\frac{AVERAGE}{\frac{AMPLITUDE}{PEAK}}\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

EDE (PUNCH) = 5

EDE (PRINT,PUNCH,AMPLITUDE) = ALL

Option	Meaning
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
AVERAGE	Calculate average energy (in frequency response only) - default
AMPLITUDE	Calculate the amplitude of the energy (in frequency response only)
PEAK	Calculate the Peak energy (for frequency response only)
ALL	Energies will be output for all elements for which stiffness matrices exist.
NONE	Energies for no elements will be output.
n	Set identification of previously appearing SET command (Integer > 0). Only strain energies for elements whose identification numbers appear on this SET command will be output.

Remarks

1. Element energy loss is available in normal modes, transient response, and frequency response analysis only.
2. This command will calculate energy loss due to viscous damping only. Energy loss due to structural damping and modal damping are not included in the results of this command.
3. The output will be in SORT1 format.
4. Both PRINT and PUNCH may be requested.
5. EDE = NONE allows overriding an overall output request.
6. The energy density output is currently calculated for CONROD, ROD, BAR, BEAM, QUAD4, TRIA3, SHEAR, HEXA, PENTA, and TETRA elements. All other element types will have the energy density set to 0.0.
7. SYSTEM(123) is used as a filter on ESE output. Any energy which is less than $10^{SYSTEM(123)}$ is set to 0.0
8. This command is stored in words 215 through 218 of CASECC.
9. AVERAGE, AMPLITUDE, and PEAK are used only in frequency response analysis.

   The solution to frequency response takes the form

$$U(t) = (u_r + iu_i)e^{i\omega t}$$

```
Where:
U(t) = displacement as a function of time
```

u_r = real component of the displacement
u_i = imaginary component of the displacement

Given this, the following three formulae are used to calculate the Element Energy Loss.

AVERAGE

$$E_{avg} = \pi\omega(\{u_r\}^T\left[Be\right]\{u_r\} + \{u_i\}^T\left[Be\right]\{u_i\})$$

AMPLITUDE:

$$E_{amp} = \pi\omega\sqrt{(\{u_r\}^T\left[Be\right]\{u_r\} + \{u_i\}^T\left[Be\right]\{u_i\})^2 + (2\{u_r\}^T\left[Be\right]\{u_i\})^2}$$

PEAK

$$E_{peak} = E_{avg} + E_{amp}$$

Where:
E = calculated element energy
[Be] = element viscous damping matrix (does not including structural damping)

EKE - *Element Kinetic Energy Output Request*

Description

Requests element kinetic energy output.

Format and Examples

$$EKE\left[\left(\frac{PRINT}{PUNCH}\right)\right]\begin{bmatrix} \frac{AVERAGE}{AMPLITUDE} \\ PEAK \end{bmatrix} = \begin{Bmatrix} ALL \\ n \\ NONE \end{Bmatrix}$$

EKE (PUNCH) = 5

EKE (PRINT,PUNCH) = ALL

Option	Meaning
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
AVERAGE	Calculate average energy (in frequency response only) - default
AMPLITUDE	Calculate the amplitude of the energy (in frequency response only)
PEAK	Calculate the Peak energy (for frequency response only)
ALL	Energies will be output for all elements for which stiffness matrices exist.
NONE	Energies for no elements will be output.
n	Set identification of previously appearing SET command (Integer > 0). Only strain energies for elements whose identification numbers appear on this SET command will be output.

Remarks

1. Element kinetic energies are available in normal modes, transient response, and frequency response only.
2. The output will be in SORT1 format.
3. Both PRINT and PUNCH may be requested.
4. EKE = NONE allows overriding an overall output request.
5. The energy density output is currently calculated for CONROD, ROD, BAR, BEAM, QUAD4, TRIA3, SHEAR, HEXA, PENTA, and TETRA elements. All other element types will have the energy density set to 0.0.
6. SYSTEM(123) is used as a filter on EKE output. Any energy which is less than $10^{SYSTEM(123)}$ is set to 0.0
7. This command is stored in words 210 through 213 of CASECC.
8. AVERAGE, AMPLITUDE, and PEAK are used only in frequency response analysis.

   The velocity in a frequency response solution takes the form

$$V(t) = (v_r + iv_i)e^{i\omega t}$$

Where:
$V(t)$ = velocity as a function of time
v_r = real component of the velocity
v_i = imaginary component of the velocity

Given this, the following three formulae are used to calculate the Element Kinetic Energy.

AVERAGE

$$E_{avg} = \frac{1}{4}(\{v_r\}^T\left[Me\right]\{v_r\} + \{v_i\}^T\left[Me\right]v\{u_i\})$$

AMPLITUDE:

$$E_{amp} = \frac{1}{4}\sqrt{\left(\left(\{u_r\}^T\left[Me\right]\{u_r\} + \{u_i\}^T\left[Me\right]\{u_i\}\right)^2 + (2\{u_r\}^T\left[Me\right]\{u_i\})^2\right)}$$

PEAK

$$E_{peak} = E_{avg} + E_{amp}$$

```
Where:
E = calculated element energy
[Me] = element mass matrix
```

ELFORCE - *Element Force Output Request*

Description

Requests form and type of element force output.

Format and Examples

$$ELFORCE\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}\right\}, \begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH,\\PLOT\end{matrix} \begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}\right)\right] = \left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

```
ELFORCE  =  ALL
ELFORCE(REAL, PUNCH, PRINT)  =  17
ELFORCE  =  25
ELFORCE(SORT2,NOPRINT)  = ALL
```

Option	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each element type. SORT2 is available only in static analysis, transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
NOPRINT	Force is calculated and saved on output file. The output file will not be sent to the output device
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Forces for all elements will be output.
NONE	Forces for no elements will be output.
n	Set identification of a previously appearing SET command. Only forces of elements whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large output files. An alternative is to define a SET of interest.
3. In static analysis or frequency response problems, any request for SORT2 output causes all output to be SORT2.
4. FORCE is an alternate form and is entirely equivalent to ELFORCE.
5. ELFORCE = NONE allows overriding an overall request.

ELSTRESS - *Element Stress Output Request*

Description

Requests form and type of element stress output.

Format and Examples

$$ELSTRESS\left[\left(\left\{\begin{matrix} SORT1 \\ SORT2 \end{matrix}\right\}, \begin{matrix} PRINT \\ \hline NOPRINT, PUNCH \\ PLOT \end{matrix}, \begin{matrix} REAL \\ \hline IMAG \\ PHASE \end{matrix}, \frac{VONMISES}{MAXSHEAR}\right)\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

```
ELSTRESS  =  5
ELSTRESS  =  ALL
ELSTRESS(SORT1, PRINT, PUNCH, PHASE) = 15
```

Option	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each element type. SORT2 is available only in static analysis, transient, and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
NOPRINT	Stresses are calculated and saved on output file. The output file will not be sent to the output device
EXTREME or LAYER	Requests stresses to be calculated at the extreme (top and bottom) fibers of a plate element or, for composites, the stresses for each layer. (See Remarks 7 and 8)
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Stresses for all elements will be output.
n	Set identification of a previously appearing SET command (Integer > 0). Only stresses for elements whose identification numbers appear on this SET command will be output.
NONE	Stresses for no elements will be output.

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large output files. An alternative is to define a SET of interest.
3. In static analysis or frequency response problems, any request for SORT2 output causes all output to be SORT2.
4. ELSTRESS is an alternate form and is entirely equivalent to STRESS.
5. ELSTRESS = NONE allows overriding an overall request.
6. The option EXTREME and LAYER is only applicable for the QUAD4 and TRIA3 elements.

ESE - *Element Strain Energy Output Request*

Description

Requests element strain energy output.

Format and Examples

$$ESE\left[\left(\frac{PRINT}{PUNCH}\right)\right]\left[\frac{AVERAGE}{\begin{matrix}AMPLITUDE\\ PEAK\end{matrix}}\right] = \left\{\begin{matrix}ALL\\ n\\ NONE\end{matrix}\right\}$$

ESE (PUNCH) = 5

ESE (PRINT,PUNCH) = ALL

Option	Meaning
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
AVERAGE	Calculate average energy (in frequency response only) - default
AMPLITUDE	Calculate the amplitude of the energy (in frequency response only)
PEAK	Calculate the Peak energy (for frequency response only)
ALL	Energies will be output for all elements for which stiffness matrices exist.
NONE	Energies for no elements will be output.
n	Set identification of previously appearing SET command (Integer > 0). Only strain energies for elements whose identification numbers appear on this SET command will be output.

Remarks

1. Element strain energies are output from static analysis. normal modes, transient response, and frequency response only.
2. The output will be in SORT1 format.
3. Both PRINT and PUNCH may be requested.
4. ESE = NONE allows overriding an overall output request.
5. The energy density output is currently calculated for CONROD, ROD, BAR, BEAM, QUAD4, TRIA3, SHEAR, HEXA, PENTA, and TETRA elements. All other element types will have the energy density set to 0.0.
6. SYSTEM(123) is used as a filter on ESE output. Any energy which is less than $10^{SYSTEM(123)}$ is set to 0.0
7. This command is stored in words 205 through 208 in CASECC.
8. AVERAGE, AMPLITUDE, and PEAK are used only in frequency response analysis.

   The solution to frequency response takes the form

$$U(t) = (u_r + iu_i)e^{i\omega t}$$

Where:
U(t) = displacement as a function of time
u_r = real component of the displacement
u_i = imaginary component of the displacement

Given this, the following three formulae are used to calculate the Element Strain Energy.

AVERAGE

$$E_{avg} = \frac{1}{4}(\{u_r\}^T[Ke]\{u_r\} + \{u_i\}^T[Ke]\{u_i\})$$

AMPLITUDE:

$$E_{amp} = \frac{1}{4}\sqrt{(\{u_r\}^T[Ke]\{u_r\} + \{u_i\}^T[Ke]\{u_i\})^2 + (2\{u_r\}^T[Ke]\{u_i\})^2}$$

PEAK

$$E_{peak} = E_{avg} + E_{amp}$$

```
Where:
E = calculated element energy
[Ke] = element stiffness matrix (not including structural damping)
```

FORCE - *Element Force Output Request*

Description

Requests form and type of element force output.

Format and Examples

$$FORCE\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}\right\}, \begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{matrix}, \begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}\right)\right] = \left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

FORCE = ALL

FORCE(REAL, PUNCH, PRINT) = 17

FORCE = 25

Option	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each element type. SORT2 is available only in static analysis, transient, and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	TThe specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary printout on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Forces for all elements will be output.
n	Set identification of a previously appearing SET command. Only forces whose element identification numbers appear on this SET command will be output (Integer > 0).
NONE	Forces for no elements will be output.

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative to this would be to define a SET of interest.
3. In static analysis or frequency response problems, any request for SORT2 output causes all output to be SORT2.
4. ELFORCE is an alternate form and is entirely equivalent to FORCE.
5. FORCE = NONE allows overriding an overall request.

FREQUENCY - *Frequency Set Selection*

Description

Selects the set of frequencies to be solved in frequency response problems.

Format and Examples

FREQUENCY = n

FREQUENCY = 17

Option	Meaning
n	Set identification of bulk data FREQ, FREQ1, FREQ2 or FREQ4 type entries (Integer > 0).

Remarks

1. The FREQ, FREQ1, or FREQ2 bulk data entries will not be used unless selected by the FREQUENCY Case Control command. Please note: A common mistake on the part of new users is to assume that the frequency set defined in bulk data will be automatically incorporated in the run ... it is not!
2. A frequency set selection is required for a frequency response problem.
3. A frequency set selection is required for transient response by Fourier methods.

GPFORCE - *Grid Point Force Balance Output Request*

Description

Requests grid point force balance output from applied loads, single-point constraints, and element constraints.

Format and Examples

$$GPFORCE\left[\left(\left\{\begin{array}{c}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{array}\right\}\right)\right] = \left\{\begin{array}{c}ALL\\n\\NONE\end{array}\right\}$$

Option	Meaning
PRINT	The printer will be the output device.
PUNCH	TThe specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
ALL	Force balance will be output for all elements connected to grid points or scalar points.
NONE	Force balance for no grid points will be output.
n	Set identification of previously appearing SET command (Integer > 0). Only force balance for points whose identification numbers appear on this SET command will be output.

Remarks

1. Grid point force balance is output from Statics Analysis (Solution Sequence 1) only.
2. The output will be in SORT1 format.
3. Both PRINT and PUNCH may be requested.
4. GPFORCE = NONE allows overriding an overall output request.

GPKE - *Grid Point Kinetic EnergyOutput Request*

Description

Requests grid point kinetic energy from normal modes.

Format and Examples

$$GPKE\left[\left(\left\{\begin{matrix}\underline{PRINT}\\ NOPRINT,PUNCH\\ PLOT\end{matrix}\right\}\right)\right]=\left\{\begin{matrix}ALL\\ n\\ NONE\end{matrix}\right\}$$

Option	Meaning
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
ALL	GPKE will be output for G grid points and scalar points.
NONE	GPKE for no grid points will be output.
n	Set identification of previously appearing SET command (Integer > 0). Only GPKE for points whose identification numbers appear on this SET command will be output.

Remarks

1. GPKE is available in Normal modes (Solution Sequence 3) only.
2. The output will be in SORT1 format.
3. Both PRINT and PUNCH may be requested.
4. GPKE = NONE allows overriding an overall output request.
5. SYSTEM cell 121 operates as a filter on the GPKE output.by changing any terms with a magnitude less than $1.x10^{SYSTEM(121)}$ to 0.0
6. The formula used to calculate GPKE is: GPKE = Φ * MΦ (where * indicates a term-by-term multiply). The scale factor of 1/2 is not included for convenience. If your modes are mass-normalized and you have used the lumped (diagonal) mass formulation, the equation provides the fraction of the total kinetic energy at each dof selected. If you have used a coupled mass formulation, the coupling in the mass matrix may cause inaccuracy in the calculation of the GPKE terms.

HARMONICS - *Harmonic Printout Control*

Description

Controls number of harmonics output for problems containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format and Examples

$$HARMONICS = \begin{Bmatrix} ALL \\ n \\ NONE \end{Bmatrix}$$

Option	Meaning
ALL	All harmonics will be output.
NONE	No harmonics will be output.
n	Available harmonics up to and including n will be output (Integer ≥ 0).

Remarks

1. If no HARMONICS command appears in Case Control, output will limited to harmonic index 0.

IC - *Transient Initial Condition Set Selection*

Description

To select the initial conditions for direct transient problems.

Format and Examples

IC = n

IC = 17

Option	Meaning
n	Set identification of a TIC bulk data entry (Integer > 0) for structural analysis.

Remarks

1. A TIC bulk data entry will not be used (hence no initial conditions) unless selected by the IC Case Control command.
2. Non-zero initial conditions are not allowed in a modal transient problem.

INCLUDE - *Directive to Read Input statements*

Description

Defines a file that contains the input commands.

Format and Examples

$$\texttt{INCLUDE,}\ \left\{\begin{array}{l}\texttt{,NOPRINT}\\ \texttt{,NOPRINT,}\\ \langle\texttt{NOPRINT}\rangle\end{array}\right\}\texttt{[=]}\quad\texttt{'filename'}$$

```
INCLUDE ‘c:\user\specialalter\myalter.alt’
INCLUDE NOPRINT ‘c:\user\specialalter\myalter.alt’
INCLUDE, NOPRINT ‘c:\user\specialalter\myalter.alt’
INCLUDE, NOPRINT, ‘c:\user\specialalter\myalter.alt’
INCLUDE (NOPRINT) ‘c:\user\specialalter\myalter.alt’
INCLUDE = ‘c:\user\specialalter\myalter.alt’
INCLUDE NOPRINT  = ‘c:\user\specialalter\myalter.alt’
INCLUDE, NOPRINT  = ‘c:\user\specialalter\myalter.alt’
INCLUDE (NOPRINT) = ‘c:\user\specialalter\myalter.alt’
```

Remarks

1. This statement can be used in Executive, Case Control, and Bulk Data Sections.
2. Input statements are saved in the file named filename.
3. Comma, equal sign, and parentheses are not allowed in filename.
4. NOPRINT allows reading in the input statements, such as the DMAP alters or restart dictionary, without printing them. The default PRINT.
5. Since this statement can also be used in the Case Control Section, an equal sign is also allowed.
6. Nested `INCLUDE` is allowed.
7. The punctuation in the pathed file name is system dependent.
8. READFILE is an alternative name for INCLUDE
9. If a relative path is provided, it will be relative the the directory containing the input file.
10. I If no path is specified, the directory will be the one containing the input file.

K2GG - *Direct Input Stiffness Matrix Selection*

Description

Selects a direct input stiffness matrix associated with the G-set of displacements.

Format and Examples

K2GG = name

K2GG = WING

K2GG = K2GG

Option	Meaning
name	The same character string assigned to a [K2GG] matrix on a DMIG or DMIAX bulk data entry.

Remarks

1. K2GG matrix input is added to the damping matrix associated with the g-set of displacements.
2. The K2GG matrix must be real and symmetric.
3. DMIG and DMIAX matrices will not be used unless selected.
4. These matrices are added into the system matrices before processing begins and will be included in all processing. Effects of these matrices will show up in the Grid Point Weight Generator output and in the processing of AUTOSPC. These matrices cannot reference extra `points (EPOINTs) and` must be symmetric and real.

K2PP direct Input Stiffness Matrix Selection

Description

Selects a direct input stiffness matrix associated with the P-set of displacements.

Format and Examples

K2PP = name
K2PP = KDMIG
K2PP = K2PP

Option	Meaning
name	The same character string assigned to a [K2PP] matrix on a DMIG or DMIAX bulk data entry.

Remarks

1. K2PP is used only in dynamic solution sequences. It is not used in static or normal modes analyses
2. DMIG and DMIAX matrices will not be used unless selected.
3. These matrices are added to the system matrices after all reductions are performed and after the solution for the real (undamped) modes. Therefore, terms from these matrices will not be able to prevent mechanisms if a reduction is being done. Also, it is possible that AUTOSPC might constrain dof which these matrices connect to, as it is performed before these matrices are added to the system.These matrices may be symmetric or square and they may also be real or complex (only for frequency response and complex eigenvalue solutions). These matrices may reference any dof in the model (including Points).

LABEL - *Output Label*

Description

Defines a label which will appear on the third heading line of each page of printer output.

Format and Examples

$LABEL = \{Any\ character\ data\}$

LABEL = SAMPLE OF A LABEL

Remarks

1. LABEL appearing at the subcase level will label output for that subcase only.
2. LABEL appearing before all subcases will label any outputs which are not subcase dependent.
3. If no LABEL is supplied, the label line will be blank.
4. LABEL information is also placed on NASTRAN plotter output as applicable.

LINE - *Data Lines Per Page*

Description

Defines the number of data lines per printed page.

Format and Examples

LINE = n

LINE = 58

Option	Meaning
n	Number of data lines per page (Integer ≥ 10).

Remarks

1. If no LINE command appears, the appropriate default is used.
2. For 11 inch paper, 50 is recommended; for 8-1/2 inch paper, 35 is the recommended r.
3. Alternatively, the number of data lines per printed page can also be defined by means of the NLINES keyword on the NASTRAN statement (see **Section 2.1**).

LOAD - *External Static Load Set Selection*

Description

Selects the external static load set to be applied to the structural model.

Format and Examples

LOAD = n

LOAD = 15

Option	Meaning
n	Set identification of at least one bulk data external load entry and hence must appear on at least one FORCE, FORCE1, FORCE2, MOMENT, MOMENT1, MOMENT2, GRAV, PLOAD, PLOAD1, PLOAD2, PLOAD4, RFORCE, PRESAX, FORCEAX, MOMAX, SLOAD, or LOAD entry (Integer > 0).

Remarks

1. Bulk data static load entries will not be used unless selected by a LOAD Case Control command.
2. A GRAV entry cannot have the same set identification number as any of the other bulk data load-type entries. If a gravity load and another type of load are to be applied they must be combined using the LOAD bulk data entry.
3. If n is the set identification number of a LOAD bulk data entry, then it must be different from the load set identification numbers of all external static load sets in bulk data.
4. LOAD is only applicable in statics, inertia relief, differential stiffness, buckling, and piece wise linear problems.
5. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement (SPC) Loads and external loads specified using DMIG bulk data entries and selected using a P2G Case Control command
6. Static, thermal, and element deformation loads must have unique set identification numbers.
7. The solution sequences that accept a static load set expect the set number to be selected by a Case Control command in a certain place with respect to subcase definitions.

M2GG - *Direct input Mass Matrix Selection*

Description

Selects a direct input mass matrix associated with the G-set of displacements.

Format and Examples

M2GG = name

M2GG = WINGMASS

M2GG = M2GG

Option	Meaning
name	The same character string assigned to a [M2GG] matrix on a DMIG or DMIAX bulk data entry.

Remarks

1. M2GG can be used to augment/define mass coefficients which are added to the finite element mass in all solution sequences.M2GG matrices are added to the mass matrix associated with the g-set.
2. The M2GG matrix must be real and symmetric.
3. PARAM,WTMASS has no effect on matrices input by the M2GG command.
4. DMIG and DMIAX matrices will not be used unless selected.
5. These matrices are added into the system matrices before processing begins and will be included in all processing. Effects of these matrices will show up in the Grid Point Weight Generator output and in the processing of AUTOSPC. These matrices cannot reference extra `points (EPOINTs) and` must be symmetric and real.

M2PP - *Direct input Mass Matrix Selection*

Description

Selects a direct input mass matrix.

Format and Examples

M2PP = name

M2PP = MDMIG

M2PP = M2PP

Option	Meaning
name	The same character string assigned to a [M2PP] matrix on a DMIG or DMIAX bulk data entry.

Remarks

1. M2PP is supported only in dynamic solution sequences.
2. DMIG and DMIAX matrices will not be used unless selected.
3. These matrices are added to the system matrices after all reductions are performed and after the solution for the real (undamped) modes. Therefore, terms from these matrices will not be able to prevent mechanisms if a reduction is being done. Also, it is possible that AUTOSPC might constrain dof which these matrices connect to, as it is performed before these matrices are added to the system.These matrices may be symmetric or square and they may also be real or complex (only for frequency response and complex eigenvalue solutions). These matrices may reference any dof in the model (including EPOINTs).

MAXLINES - *Maximum Number of Output Lines*

Description

Sets the maximum number of output lines to a given value.

Format and Examples

$$MAXLINES\ =\ \left\{\frac{200000}{n}\right\}$$

MAXLINES = 50000

Option	Meaning
n	Maximum number of output lines (Integer > 0).

Remarks

1. Any time this number is exceeded, NASTRAN will terminate through PEXIT.
2. This command may or may not override system operating control specification. Check with the local operations staff.
3. Default is MAXLINES = 2^32-1

MEFFMASS - *Modal effective mass output request*

Description

Requests the calculation and output of modal effective mass, participation factors, modal effective weight in modal solutions.

Format and Examples

$$MEFFMASS\begin{bmatrix} PRINT \\ PLOT \end{bmatrix}\begin{bmatrix} ALL, SUMMARY, PARTFAC \\ FRACSUM, MEFFW, MEFFM \end{bmatrix} = \begin{Bmatrix} YES \\ NO \end{Bmatrix}$$

MEFFMASS(ALL) = YES
MEFFMASS = YES

Option	Meaning
PRINT	Write selected output to the print file. (default)
PLOT	Calculate, but do not print selected items.
ALL	Print all output related to this command
SUMMARY	Requests the total modal effective mass and reference mass (default)
PARTFAC	Requests modal participation factors
FRACSUM	Requests modal effective mass fractions and cumulative values
MEFFW	Requests modal effective weight
MEFFM	Requests modal effective mass.

Remarks

1. Modal participation factors are defined as:

   ε = participation factor = $\Phi^T M R_b$

   where:

   Φ = the calculated mode shapes (mass normalized)

   M = the mass matrix

   R_b = rigid-body vectors calculated about PARAM,GRDPNT
2. .Modal effective mass is defined as:ε^2
3. Modal effective weight is defined as the modal effective mass multiplied by the acceleration of gravity, or

   MEFFW = ε^2/WTMASS
4. Modal effective mass fractions are defined as the modal effective mass divided by the available mass in the associated direction.
5. This command is stored in words 200 thru 204 of CASECC.

METHOD - *Real Eigenvalue Extraction Method Selection*

Description

Selects the real eigenvalue parameters to be used during eigenvalue calculations.

Format and Examples

METHOD = n

METHOD = 33

Option	Meaning
n	Set identification number of an bulk data EIGR or EIGRL entry for normal modes or modal formulation or an EIGB entry for buckling, SOL 5. (Integer > 0).

Remarks

1. An eigenvalue extraction method must be selected when extracting real eigenvalues using functional module READ.
2. Each of the solution sequences that incorporate real eigenvalue extraction require an appropriate bulk data entry that is selected by a METHOD Case Control command in an appropriate place with respect to subcase definitions.

MODALSE - *Request Modal Contributions in Frequency Response*

Description

Requests modal contributions in a modal frequency response solution.

Format and Examples

MODALSE = n
MODALSE = ALL

Option	Meaning
n	Set of modes to perform the calculation on (Integer > 0).

Remarks

1. This command will only work in modal frequency response analysis.
2. The command uses words 194-196 in CASECC to store the request:

MODES - *Case Control Do-Loop*

Description

Repeats a case control output packet MODES times, to allow control of output in eigenvalue problems.

Format and Examples

MODES = n

MODES = 1

Option	Meaning
n	Number of modes, starting with the first and proceeding sequentially upward, for which the case control or subcase control is to apply. (Integer > 0).

Remarks

1. This command can best be illustrated by an example. Suppose stress output is desired for the first five modes only and displacements only thereafter. The following example would accomplish this:

```
SUBCASE 1
MODES = 5
OUTPUT
STRESS = ALL
SUBCASE 6
OUTPUT
DISPLACEMENTS = ALL
BEGIN BULK
```

2. The MODES command causes the results for each eigenvalue to be considered as a separate, successively numbered subcase, beginning with the subcase number containing the MODES command.
3. If the MODES command is not used, eigenvalue results are considered to be a part of a single subcase. Hence, any output requests for the single subcase will apply for all eigenvalues.
4. All eigenvectors with mode numbers greater than the number of records in Case Control are printed with the descriptors of the last Case Control record. For example, to suppress all printout for modes beyond the first three, the following Case Control deck could be used:

```
SUBCASE 1
MODES = 3
DISPLACEMENTS = ALL
SUBCASE 4
DISPLACEMENTS = NONE
BEGIN BULK
```

MODESELECT - *Select Modes for inclusion/exclusion from the subsequent analysis*

Description

Selects a subset of the calculated modes for inclusion/exclusion from the subsequent dynamic analysis.

Format and Examples

$$MODESELECT = \{n\}$$

SET 10 = 1,5,6
MODESELECT = 10 $ only modes 1, 5, and 6 will be used for subsequent analysis

SET 25 = 1, 10.9, 4
MODESEL = 25 $ modes 1, 4, and the mode with natural frequency closest to 10.9 cps will be used

SET 15 = 1,5,10
MODESELECT = -15 $ exclude modes 1, 5, and 10 in subsequent analysis - all other calculated modes will be included.

Option	Meaning
n	Set identification of previously appearing SET command (Integer > 0). This set will be used in the selection of the modes to include/exclude. If *n* is positive, the selected set of modes is included in subsequent analysis (all other modes are excluded). If *n* is negative, the selected set of modes is excluded from subsequent analysis.

Remarks

1. MODESELECT is defaulted to all modes if the command is not supplied.
2. Acceptable forms of this command are MODESELECT or MODESEL.
3. This command is stored in word 198 of CASECC.
4. The SET selected by this command may contain integer and/or real values. Integer values represent the selected mode number. Real values represent the natural frequency (cps) of the mode.
5. For each real number in the selected SET, the mode whose natural frequency (cps) is closest to the value is selected. NOTE that there is no checking to see if a mode is selected more than once. If the value is larger than the highest natural frequency found, the mode with the highest natural frequency will be selected. If the value is less than the lowest natural frequency found, the mode with the lowest natural frequency will be selected.
6. If an integer value in the SET is higher than the number of modes found, it will be ignored.
7. This command is used only in the modal dynamic solutions and the real eigenvalue solutions. In other solutions, it will be ignored.
8. In the real eigenvalue solutions, this command may be used to select the set of modes used for data recovery.
9. PARAMs HFREQ,LFREQ, and LMODES are independent from this command. That is, if they are entered, they will be used on the set of modes resulting from this command.
10. MODESELECT is performed after the calculation of residual vectors, so care must be excercised to be sure that residual vectors are not removed unintentionally.

11.As the default is to include all calculated modes, ALL is not acceptable as input for this command.

MPC - *Multipoint Constraint Set Selection*

Description

Selects the multipoint constraint set to be applied to the structural model.

Format and Examples

MPC = n

MPC = 17

Option	Meaning
n	Set identification of a multipoint constraint set and hence must appear on at least one MPC, MPCADD, MPCAX, or MPCS bulk data entry. (Integer > 0).

Remarks

1. MPC, MPCADD, MPCAX, or MPCS bulk data entries will not be used unless selected by an MPC Case Control command.

MPCFORCE - *Multipoint Forces of Constraint Output Request*

Description

Requests multipoint force of constraint vector output.

Format and Examples

$$MPCFORCE\left[\left(\left\{SORT1, \begin{matrix} \underline{PRINT} \\ NOPRINT,PUNCH \\ PLOT \end{matrix}\right\}\right)\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

MPCFORCE = 10

MPCFORCE(PRINT,PUNCH) = ALL

MPCFORCE = NONE

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each subcase or frequency, depending on the solution sequence. SORT2 is not available.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
ALL	Multipoint forces of constraint for all points will be output (only nonzero entries).
NONE	Multipoint forces of constraint for no points will be output.
n	Set identification of previously appearing SET command. Only multipoint constraint forces for points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. MPCFORCE = NONE allows overriding an overall output request.
3. MPCFORCE is only valid for statics and real eigenvalue analyses.
4. A request for MPCFORCE is not allowed for axisymmetric elements.
5. See the PARAM bulk data entry for use of related parameters OPT and GRDEQ.

NCHECK - *Stress and Element Forces Numerical Accuracy Check*

Description

Requests stress and element force numerical accuracy check.

Format and Examples

NCHECK [= n]

NCHECK

NCHECK = 6

Option	Meaning
n	A printout of the number of significant digits accuracy is issued for each element having an entry with less than n significant digits in the stress or force calculation.

Remarks

1. All the elements requested on the STRESS and/or FORCE command (or their equivalent ELSTRESS and/or ELFORCE command) are checked.
2. The default for n is five (5) when n is not specified.
3. These checks measure the quality of the computations to obtain element stresses and element forces. They do not measure the quality of the model being analyzed.
4. The printout identifies the element types, identification number and the subcase. The entries checked are as follows:

Element Type	Entries
ROD, CONROD, TUBE	FA, T, ε_A ε_T
BAR	FA, T, M_{1a}, M_{1b}, M_{2a}, M_{2b}, V_1, V_2, ε_a
TRIA3, QUAD4	ε_{x1}, ε_{y1}, ε_{xy1}, ε_{x2}, ε_{y2}, τ_{xy2}, M_x, M_y, M_{xy}, V_x, V_y, V_{xy}
HEXA, TETRA and PENTA	ε_x, ε_y, ε_z, τ_{yz}, τ_{xz}, τ_{xy}
SHEAR	ε_{MAX}, ε_{AVE}, corner forces, kick forces, and shears.

NLLOAD - *Nonlinear Load Output Request*

Description

Requests form and type of nonlinear load output for transient problems.

Format and Examples

$$LLOAD\left[\left(\left\{\begin{array}{c}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{array}\right\}\right)\right] = \left\{\begin{array}{c}ALL\\n\\NONE\end{array}\right\}$$

NLLOAD = ALL

Option	Meaning
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
ALL	Nonlinear loads for all solution points will be output.
NONE	Nonlinear loads will not be output.
n	Set identification of previously appearing SET command.(Integer > 0). Only non-linear loads for points whose identification numbers appear on this SET command will be output.

Remarks

1. Both PRINT and PUNCH may be used.
2. Nonlinear loads are output only in the solution (D or H) set.
3. The output format will be SORT2.
4. An output request for ALL in transient response problems generally produce large amounts of printout. An alternative to this would be to define a SET of interest.

NONLINEAR - *Nonlinear Load Set Selection*

Description

Selects nonlinear load for transient problems.

Format and Examples

NONLINEAR = *n*

NONLINEAR LOAD SET = 75

Option	Meaning
n	Set identification of NOLINi bulk data entries (Integer > 0).

Remarks

1. NOLINi bulk data entries will not be used unless selected by a NONLINEAR Case Control command.

OFREQUENCY - *Output Frequency Set*

Description

Selects from the solution set of frequencies a subset for output requests in direct or modal frequency analysis. In flutter analysis, it selects a subset of velocities.

Format and Examples

$$OFREQUENCY = \begin{Bmatrix} ALL \\ n \end{Bmatrix}$$

OFREQUENCY = ALL

OFREQUENCY SET = 15

Option	Meaning
ALL	Output for all frequencies will be printed out.
n	Set identification of previously appearing SET command (Integer > 0). Output for frequencies closest to those given on this SET command will be produced.

Remarks

1. OFREQUENCY is defaulted to ALL if it is not supplied.
2. In flutter analysis, the selected set lists velocities in input units. If there are n velocities in the list, the n points with velocities closest to those in the list will be selected for output.
3. This command is used in conjunction with the MODACC module to limit the frequencies for which mode acceleration computations are performed.

OLOAD - *Applied Load Output Request*

Description

Requests form and type of applied load vector output.

Format and Examples

$$OLOAD\left[\left(\left\{SORT1,\ SORT2,\ \frac{PRINT}{NOPRINT,PUNCH},\ PLOT,\ \frac{REAL}{IMAG},\ PHASE\right\}\right)\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

OLOAD = ALL SLOAD(SORT1, PHASE) = 5

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each grid point. SORT2 is available only in static analysis, transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Applied loads for all points will be output. (SORT1 will only output nonzero values.)
NONE	Applied loads for no points will be output.
n	Set identification of previously appearing SET command. Only loads on points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. In static analysis or frequency response problems, any request for SORT2 output causes all output to be SORT2.
4. A request for SORT2 causes loads (zero and nonzero) to be output.
5. OLOAD = NONE allows overriding an overall output request.

OTIME - *Output Time Set*

Description

Selects from the solution set of times a subset for output requests.

Format and Examples

$$OTIME = \begin{Bmatrix} ALL \\ n \end{Bmatrix}$$

OTIME = ALL
OTIME = 15

Option	Meaning
ALL	Output for all times will be printed out.
n	Set identification of previously appearing SET command. (Integer > 0). Output for times closest to those given on this SET command will be output.

Remarks

1. OTIME is defaulted to ALL if it is not supplied.
2. The OTIME command is particularly useful for restarts to request a subset of the output (that is, stresses at only peak times, etc.)
3. This command can be used in conjunction with the MODACC module to limit the times for which mode acceleration computations are performed.

OUTPUT - *Output Packet Delimiter*

Description

Delimits the various output packets, structure plotter, curve plotter, and printer/punch.

Format and Examples

$$OUTPUT\left[\left(\begin{Bmatrix} PLOT \\ POST \\ XYOUT \\ XYPLOT \end{Bmatrix}\right)\right]$$

OUTPUT(PLOT)
OUTPUT(XYOUT)

Option	Meaning
No qualifier	Beginning of printer output packet. This is not a required command.
PLOT	Beginning of structure plotter packet. This command must precede all structure plotter control commands.
XYOUT or XYPLOT	Beginning of curve plotter packet. This command must precede all curve plotter control commands. XYPLOT and XYOUT are entirely equivalent.

Remarks

1. The structure plotter packet and the curve plotter packet must be at the end of the Case Control section. Either may come first.
2. The delimiting of a printer packet is completely optional.

P2G - *Direct Input Load Matrix Selection*

Description

Selects direct input load matrices (DMIG).

Format and Examples

P2G = name

P2G = EXTERNAL

Option	Meaning
name	Name of a $\{P_g^2\}$ (Character > 0).

Remarks

1. Terms are added to the load matrix before constraints are applied
2. Matrix must be rectangular in form.
3. The number of columns in this matrix must be equal to the number of static SUBCASEs.
4. These matrices are added into the system matrices before processing begins and will be included in all processing. These matrices cannot reference extra `points (EPOINTs)`.

PLCOEFFICIENT - *Piece Wise Linear Coefficient Set*

Description

Selects the coefficient set for piece wise linear problems.

Format and Examples

PLCOEFFICIENT = n

PLCOEFFICIENT = 25

Option	Meaning
n	Set identification of PLFACT command (Integer > 0).

Remarks

1. PLFACT bulk data entries will not be used unless selected by a PLCOEFFICIENT Case Control command.

PLOTID - *Plotter Identification*

Description

Defines a character string which will appear on the first frame of any plotter output.

Format and Examples

PLOTID = {*Any character data*}

PLOTID = RETURN TO B.J. SMITH, ROOM.201, BLDG 85, ABC COMPANY

Remarks

1. PLOTID must appear before any OUTPUT(PLOT), OUTPUT(XYOUT), or OUTPUT(XYPLOT) commands.
2. The presence of PLOTID causes a special header frame to be plotted with the supplied identification plotted several times. This allows for easy identification of the NASTRAN plotter output.
3. If no PLOTID command appears, no ID frame will be plotted.
4. The PLOTID header frame will not be generated for table plotters.

RANDOM - *Random Analysis Set Selection*

Description

Selects the RANDPS and RANDTi bulk data entries to be used in random analysis.

Format and Examples

RANDOM = n

RANDOM = 177

Option	Meaning
n	Set identification of RANDPS and RANDTi bulk data entries to be used in random analysis (Integer > 0).

Remarks

1. RANDPS bulk data entries must be selected to do random analysis.
2. RANDPS bulk data entries must be selected in the first subcase of the current loop by a RANDOM Case Control command. RANDPS may not reference subcases in a different loop.

READFILE - *Directive to Read Input statements*

Description

Defines a file that contains the input commands.

Format and Examples

```
READFILE, { PRINT,    } [ = ]  'filename'
          { <NOPRINT> }
```

(PRINT, is underlined as the default.)

```
READFILE 'c:\user\specialalter\myalter.alt'
READFILE NOPRINT 'c:\user\specialalter\myalter.alt'
READFILE, NOPRINT 'c:\user\specialalter\myalter.alt'
READFILE, NOPRINT, 'c:\user\specialalter\myalter.alt'
READFILE (NOPRINT) 'c:\user\specialalter\myalter.alt'
READFILE = 'c:\user\specialalter\myalter.alt'
READFILE NOPRINT  = 'c:\user\specialalter\myalter.alt'
READFILE, NOPRINT  = 'c:\user\specialalter\myalter.alt'
READFILE (NOPRINT) = 'c:\user\specialalter\myalter.alt'
```

Remarks

1. This statement can be used in Executive, Case Control, and Bulk Data Sections.
2. Input statements are saved in the file named filename.
3. Comma, equal sign, and parentheses are not allowed in filename.
4. NOPRINT allows reading in the input statements, such as the DMAP alters or restart dictionary, without printing them. The default PRINT.
5. Since this statement can also be used in the Case Control Section, an equal sign is also allowed.
6. Nested `READFILE` is allowed.
7. The punctuation in the pathed file name is system dependent.
8. READFILE is an alternative name for INCLUDE

REPCASE - *Repeat Case Subcase Delimiter*

Description

Delimits and identifies a repeated subcase.

Format and Examples

REPCASE n

REPCASE 137

Option	Meaning
n	Subcase identification number (integer > 1).

Remarks

1. The subcase identification number, n, must be strictly increasing (that is, greater than all previous subcase identification numbers).
2. This case will only re-output the previous real case. This allows additional set specification.
3. REPCASE may only be used in statics or inertia relief.
4. One or more repeated subcases (REPCASEs) must immediately follow the subcase (SUBCASE) to which they refer (see **example 4 in Section 2.3.3**).

SACCELERATION - *Solution Set Acceleration Output Request*

Description

Requests form and type of solution set acceleration output.

Format and Examples

$$SACCELERATION\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}\right\}, \begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{matrix}, \begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}\right)\right] = \left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

SACCELERATION = ALL

SACCELERATION(PUNCH, IMAG) = 142

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number). SORT2 is available only in transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on frequency response problems.
ALL	Acceleration for all solution points (modes) will be output.
NONE	Acceleration for no solution points (modes) will be output.
n	Set identification of a previously appearing SET command. Only accelerations of points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. Acceleration output is only available for transient and frequency response problems.
4. In a frequency response problem any request for SORT2 output causes all output to be SORT2.
5. SACCELERATION = NONE allows overriding an overall output request.

SCAN - *Output Scan Request*

Description

Scan output data and eliminate values that do not meet the specification set by this SCAN command.

Format and Examples

$$SCAN\left\{\left(\left\{\begin{matrix} STRESS \\ FORCE \\ DISP \\ LOAD \\ SPC \\ HELP \end{matrix}\right\}, data, component\right)\right\} = \left\{\begin{matrix} topn \\ max, min \end{matrix}\right\}[,SET \quad i]$$

SCAN (STRESS, CBAR, AXIAL) = 10
SCAN (STRESS, BAR, AXIAL, SA-MAX) = 15, SET 102
SCAN (FORCE, ROD, 2, 3) = 17
SCAN (FORCE, 3, CROD, 2) = +2000., -1500., SET 102
SCAN (ROD, AXIAL, FORCE, TORQUE) = 5000., 400.
SCAN (HELP)

Option	Meaning
STRESS	Request scan on stress file, of SORT1 or SORT2 format.
FORCE	Request scan on force file, of SORT1 or SORT2 format.
element	Any NASTRAN element name, with or without the leading letter "C" (BCD).
component	One or more components specified by keywords (BCD), or by numeric codes (Integer > 0). The numeric codes are the field numbers on the heading of the output page, whose values are to be scanned. (Each element has its own page heading.) See Remark 11 for the keywords and their corresponding field numbers.
topn	The highest n values, and the lowest n values, found by SCAN in the field(s) specified by component are printed out; for example, top n tension and top n compression stresses (Integer > 0).
max, min	Values greater than max and less than min, in the field(s) specified by component, are printed out (Real).
i	Element set identification of a previously appearing SET command (Integer > 0). Only forces or stresses of elements whose identification numbers appear on this SET command will be scanned for output. (Default is all.)
HELP	A table of the component keywords and their corresponding field numbers will be printed immediately before the Bulk Data Deck, and the job will continue.
ON-LINE	Request SCAN operation to be run on-line under real-time environment.

Remarks

1. Multiple SCAN commands can be requested. They do not override one another.
2. A SCAN command specifies only one element type; an element type can have more than one SCAN command.
3. More than one component field can be requested in a SCAN command. However, these fields will be scanned together as a group.
4. SCAN sorts and prints the scanned values in descending order. All fields of the same output line are printed.

5. If the component keyword is misspelled, a list of the valid names and their corresponding fields will be printed automatically and the job will be flagged for fatal error termination.
6. Some component keywords imply multi-field scan; for example, "AXIAL" may imply axial forces for grid points 1, 2, 3, etc.
7. Component numeric code specifies field numbers 1 through 62 only.
8. Normally, SCAN will scan only data already generated for the Output File Processor (OFP). That is, SCAN cannot scan data that has not been created. However, if no ELSTRESS (or STRESS) command is specified before a stress SCAN command, a STRESS command is generated internally in the following form:

```
STRESS (SORT1, NOPRINT, REAL) = ALL
```

   Forces are handled similarly.
9. The LABEL line (after TITLE and SUBTITLE) is limited to 36 characters. The rest of the line is replaced by the SCAN header.
10. When the ON-LINE option is requested, the other input parameters are not needed on the SCAN command. These parameters will be prompted for on the CRT screen by the computer system when the SCAN module is executed.
11. The component keywords for stress and force, and their corresponding output field numbers, are listed below. This table and the SCAN capability needs to be checked out.

Force/Stress	Keyword	Component (Output Field No.)
ROD, TUBE, CONROD		
STRESS	AXIAL	2
STRESS	TORSIONAL	4
STRESS	MARGIN	3, 5
FORCE	AXIAL	2
FORCE	TORQUE	3
SHEAR, TWIST		
STRESS	MAX-SHR	2
STRESS	MARGIN	4
STRESS	AVG	3
STRESS	MAX	2
FORCE	FORCE-1	2
FORCE	FORCE-2	3
FORCE	MOMENT-1	2
FORCE	MOMENT-2	3
TRIA1, TRIA2, QUAD1, QUAD2, TRBSC, TRPLT, QDPLT		
STRESS	NORM-X	3, 11
STRESS	NORM-Y	4, 12
STRESS	SHEAR-XY	5, 13
STRESS	MAJOR	7, 15
STRESS	MINOR	8, 16
STRESS	MAX-SHR	9, 17
FORCE	MOMENT-X	2
FORCE	MOMENT-Y	3
FORCE	SHEAR-X	5
FORCE	SHEAR-Y	6
FORCE	TWIST	4
TRMEM, QDMEM, QDMEM1, QDMEM2		
STRESS	NORM-X	2
STRESS	NORM-Y	3
STRESS	SHEAR-XY	4
STRESS	MAJOR	6
STRESS	MINOR	7
STRESS	MAX-SHR	8
FORCE	FORCE-12	3, 4
FORCE	FORCE-23	5, 6

Force/Stress	Keyword	Component (Output Field No.)
FORCE	FORCE-34	7, 8
FORCE	FORCE-41	2, 9
FORCE	KICK ON1	10
FORCE	KICK ON2	12
FORCE	KICK ON3	14
FORCE	KICK ON4	16
FORCE	SHEAR-XY	11
FORCE	SHEAR-YZ	13
FORCE	SHEAR-ZX	15
FORCE	SHEAR	17
ELAS1, ELAS2, ELAS3, IS2D8		
STRESS	OCT-SHR	2
FORCE	CIRCUM	2
FORCE	FORCE-1	4, 9
FORCE	FORCE-2	3, 6
FORCE	FORCE-3	5, 8
FORCE	FORCE-4	2, 7
BAR, ELBOW		
STRESS	SA-MAX	7, 8
STRESS	SB-MAX	14, 15
STRESS	MARGIN	9, 16
STRESS	AXIAL	6
FORCE	AXIAL	8
FORCE	TORQUE	9
FORCE	SHEAR	5, 6
FORCE	MOMENT-A	2, 3
FORCE	MOMENT-B	4, 5
CONEAX		
STRESS	NORM-U	4, 22
STRESS	NORM-V	5, 23
STRESS	SHEAR-UV	6, 24
STRESS	MAJOR	8, 26
STRESS	MINOR	9, 27
STRESS	MAX-SHR	10, 28
FORCE	MOMENT-U	3
FORCE	MOMENT-V	4
FORCE	SHEAR-XY	6
FORCE	SHEAR-YZ	7
TRIARG		
STRESS	RADIAL	2
STRESS	CIRCUM	3
STRESS	AXIAL	4
STRESS	SHEAR	5
FORCE	RADIAL	2, 5, 8
FORCE	CIRCUM	3, 6, 9
FORCE	AXIAL	4, 7, 10
TRAPRG		
STRESS	RADIAL	2,
STRESS	CIRCUM	3,
STRESS	AXIAL	4,
STRESS	SHEAR	5,
STRESS	SHR-FBRC	6, 10, 14, 18 ... 26
FORCE	RADIAL	2,
FORCE	CIRCUM	3,
FORCE	AXIAL	4,
TORDRG		
STRESS	MEM-T	2,

Force/Stress	Keyword	Component (Output Field No.)
STRESS	MEM-C	3,
STRESS	FLEX-T	4,
STRESS	FLEX-C	5, 10, 15
STRESS	SHR-FORC	6, 11, 16
FORCE	RADIAL	2,
FORCE	CIRCUM	3,
FORCE	AXIAL	4, 10
FORCE	MOMENT	5, 11
FORCE	CURV	7, 13
IHEX1, IHEX2		
STRESS	NORM-X	3, 25, 47, 69 ... etc.
STRESS	SHEAR-XY	4, 26, 48, 70 ... etc.
STRESS	PRINC-A	5, 27, 49, 71 ... etc.
STRESS	MEAN	9, 31, 53, 75 ... etc.
STRESS	NORM-Y	11, 33, 55, 77 ... etc.
STRESS	SHEAR-YZ	12, 34, 56, 78 ... etc.
STRESS	PRINC-B	13, 35, 57, 79 ... etc.
STRESS	NORM-Z	17, 39, 61, 83 ... etc.
STRESS	SHEAR-ZX	18, 40, 62, 84 ... etc.
STRESS	PRINC-C	19, 41, 63, 85 ... etc.
STRESS	MAX-SHR	10, 32, 54, 76 ... etc.
STRESS	OCT-SHR	10, 32, 54, 76 ... etc.
IHEX3		
STRESS	NORM-X	3, 26, 49, 72 ... 739
STRESS	SHEAR-XY	4, 27, 50, 73 ... 740
STRESS	PRINC-A	5, 28, 51, 74 ... 741
STRESS	MEAN	9, 32, 55, 78 ... 745
STRESS	NORM-Y	12, 35, 58, 81 ... 748
STRESS	SHEAR-YZ	13, 36, 59, 82 ... 749
STRESS	PRINC-B	14, 37, 60, 83 ... 750
STRESS	NORM-Z	18, 41, 64, 87 ... 754
STRESS	SHEAR-ZX	19, 42, 65, 88 ... 755
STRESS	PRINC-C	20, 43, 66, 89 ... 756
STRESS	MAX-SHR	10, 33, 56, 79 ... 746
STRESS	OCT-SHR	10, 33, 56, 79 ... 746
TRIAAX, TRAPAX		
STRESS	RADIAL	3, 11, 19
STRESS	AXIAL	4, 12, 20
STRESS	CIRCUM	5, 13, 21
STRESS	MEM-C	6, 14, 22
STRESS	FLEX-T	7, 15, 23
STRESS	FLEX-C	8, 16, 24
FORCE	RADIAL	3,
FORCE	CIRCUM	4,
FORCE	AXIAL	5,
QUAD4, TRIA3		
STRESS	NORMAL-X	3, 11
STRESS	NORMAL-Y	4, 12
STRESS	SHEAR-XY	5, 13
STRESS	MAJOR	7, 15
STRESS	MINOR	18, 16
STRESS	MAX-SHR	9, 17
FORCE	FX+FY	2,
FORCE	FXY	4
FORCE	MX+MY	5,
FORCE	MXY	7
FORCE	VX+VY	8, 19

Force/Stress	Keyword	Component (Output Field No.)
STRESS	NORMAL-1	5, 15, 25, 35
STRESS	NORMAL-2	6, 16, 26, 36
STRESS	SHEAR-12	7, 17, 27, 37
STRESS	SHEAR-1Z	10, 20, 30, 40
STRESS	SHEAR-2Z	11, 21, 31, 41

Use output field number(s) to specify component(s) for elements or keywords not listed above. See **sections 2.3.51 and 2.3.52 of the Programmer's Manual** for additional element stress and force component definitions.

SDAMPING - *Structural Damping*

Description

Selects table which defines damping as a function of frequency in modal formulation problems.

Format and Examples

SDAMPING = n

SDAMPING = 77

Option	Meaning
n	Set identification of a TABDMP1 table (Integer > 0).

Remarks

1. In NASTRAN-xMG, damping is cumulative. This means that any damping applied by the SDAMPING command will be combined with any other damping defined in the problem.

SDISPLACEMENT - *Solution Set Displacement Output Request*

Description

Requests form and type of solution set displacement output.

Format and Examples

$$SDISPLACEMENT\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}, \begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{matrix}, \begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}\right\}\right)\right] = \left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

SDISPLACEMENT = ALL

SDISPLACEMENT(SORT2, PUNCH, PHASE) = NONE

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number). SORT2 is available only in transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Displacements for all points (modes) will be output.
NONE	Displacements for no points (modes) will be output.
n	Set identification of previously appearing SET command. Only displacements of points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. In a frequency response problem any request for SORT2 causes all output to be SORT2.
4. SVECTOR is an alternate form which is entirely equivalent to SDISPLACEMENT.
5. SDISPLACEMENT = NONE allows overriding an overall output request.

SET - *Set Definition*

Description

1. Lists identification numbers (point or element) for output requests.
2. Lists the frequencies for which output will be printed in frequency response problems.

Format and Examples

1. Defining sets of elements or points

   SET n = {i1 i2, i3 THRU i4 EXCEPT i5, i6, i7, i8 THRU i9]}

   SET 77 = 5

   SET 88 = 5, 6, 7, 8, 9, 10 THRU 55 EXCEPT 15, 16, 77, 78, 79, 100 THRU 300

   SET 99 = 1 THRU 100000

2. Defining sets of frequencies

   SET n = {r1[, r2, r3, r4]}

   SET 101 = 1.0, 2.0, 3.0

   SET 105 = 1.009, 10.2, 13.4, 14.0, 15.0

Option	Meaning
n	Set identification (Integer > 0). Any set may be redefined by reassigning its identification number. Sets inside SUBCASE delimiters are local to the SUBCASE.
i1, i2 etc.	Element or point identification number at which output is requested. (Integer > 0) If no such identification number exists, the request is ignored.
i3 THRU i4	Output at set identification numbers i3 through i4 (i4 > i3).
EXCEPT	Set identification numbers following EXCEPT will be deleted from output list as long as they are in the range of the set defined by the immediately preceding THRU.
r1, r2 etc.	Frequencies for output (Real ≥ 0.0). The nearest solution frequency will be output. EXCEPT and THRU cannot be used.

Remarks

1. A SET command may be more than one physical command. A comma (,) at the end of a input line signifies a continuation will follow. Commas may not end a set.
2. Identification numbers following EXCEPT within the range of the THRU must be in ascending order.
3. In the first format, i8 must be greater than i4; that is, the THRU must not be within an EXCEPT range.
4. The SET cannot be ALL.

SPC - *Single-Point Constraint Set Selection*

Description

Selects the single-point constraint set to be applied to the structural model.

Format and Examples

SPC = n

SPC = 10

Option	Meaning
n	Set identification of a single-point constraint set and hence must appear on an SPC, SPC1, SPCADD, SPCAX, SPCS, or SPCS1 bulk data entries (Integer > 0).

Remarks

1. SPC, SPC1, SPCADD, SPCAX, SPCS, or SPCS1 bulk data entries will not be used unless selected by an SPC Case Control command.

SPCFORCES - *Single-Point Forces of Constraint Output Request*

Description

Requests form and type of single point force of constraint vector output.

Format and Examples

$$SPCFORCES\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}\right\},\begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{matrix},\left\{\begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}\right\}\right)\right]=\left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

SPCFORCES = 5

SPCFORCES(SORT2, PUNCH, PRINT, IMAG) = ALL

SPCFORCES(PHASE) = NONE

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each grid point. SORT2 is available only in static analysis, transient, and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Single point forces of constraint for all points will be output. (SORT1 will only output nonzero values.)
NONE	Single point forces of constraint for no points will be output.
n	Set identification of previously appearing SET command. Only single-point forces of constraint for points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. In static analysis or frequency response problems, any request for SORT2 output causes all output to be SORT2.
4. A request for SORT2 causes loads (zero and nonzero) to be output.
5. SPCFORCES = NONE allows overriding an overall output request.

STRAIN - *Element Strain/Curvature Output Request*

Description

Requests element strain/curvature output.

Format and Examples

$$STRAIN\left[\left(\left\{\frac{PRINT}{NOPRINT}, PUNCH, \frac{VONMISES}{MAXSHEAR}, \frac{STRCUR}{FIBER} \atop PLOT\right\}\right)\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

STRAIN (PUNCH) = 5

STRAIN (PRINT,PUNCH) = ALL

Option	Meaning
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
ALL	Strains/curvatures for all elements will be output. See Remark 5.
NONE	Strains/curvatures for no elements will be output.
n	Set identification of previously appearing SET command (Integer > 0). Only strains/curvatures for elements whose identification numbers appear on this SET command will be output. See Remark 5.

Remarks

1. Element strains/curvatures are output from static analysis (Solution Sequence 1) only.
2. The output will be in SORT1 format.
3. Both PRINT and PUNCH may be requested.
4. STRAIN = NONE allows overriding an overall output request.
5. Strains/curvatures are computed only for TRIA3and QUAD4 elements.
6. The format of the two-line output for each element consists of strain in the middle surface (line 1) and curvature (line 2).

STRESS - *Element Stress Output Request*

Description

Requests form and type of element stress output.

Format and Examples

$$STRESS\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}, \begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{matrix}, \begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}, \begin{matrix}\underline{VONMISES}\\MAXSHEAR\end{matrix}\right\}\right)\right] = \left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

```
STRESS = 5
STRESS = ALL
STRESS(SORT1, PRINT, PUNCH, PHASE) = 15
```

Option	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each element type. SORT2 is available only in static analysis, transient, and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
NOPRINT	Stresses are calculated and saved on file which is not sent to output device.
EXTREME or LAYER	Requests stresses to be calculated at the extreme (top and bottom) fibers of a plate element or, for composites, the stresses for each layer. (See Remarks 7 and 8.)
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Stresses for all elements will be output.
n	Set identification of a previously appearing SET command (Integer > 0). Only stresses for elements whose identification numbers appear on this SET command will be output.
NONE	Stresses for no elements will be output.

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. In static analysis or frequency response problems, any request for SORT2 output causes all output to be SORT2.
4. STRESS is an alternate form and is entirely equivalent to ELSTRESS.
5. STRESS = NONE allows overriding an overall request.
6. When LAYER is selected, individual layer stresses and/or failure indices will be output.

7. The options EXTREME and LAYER are only applicable for the QUAD4 and TRIA3 elements.

SUBCASE - *Subcase Delimiter*

Description

Delimits and identifies a subcase.

Format and Examples

SUBCASE n

SUBCASE 101

Option	Meaning
n	Subcase identification number (Integer > 0).

Remarks

1. The subcase identification number, n, must be strictly increasing (that is, greater than all previous subcase identification numbers).
2. Plot requests and RANDOM requests refer to n.

SUBCOM - *Combination Subcase Delimiter*

Description

Delimits and identifies a combination subcase.

Format and Examples

SUBCOM n

SUBCOM 125

Option	Meaning
n	Subcase identification number (Integer > 2).

Remarks

1. The subcase identification number, n, must be strictly increasing (that is, greater than all previous subcase identification numbers).
2. A SUBSEQ command may appear in this subcase.
3. SUBCOM may only be used in statics or inertia relief problems.
4. Output requests above the subcase level will be utilized.
5. Up to 360 SUBCOM commands can be used in one analysis.

SUBSEQ - *Subcase Sequence Coefficients*

Description

Gives the coefficients for forming a linear combination of the previous subcases.

Format and Examples

$SUBSEQ = R_1[,R_2, R_3, \ldots, R_n]$

SUBSEQ = 1.0, -1.0, 0.0, 2.0

Option	Meaning
R1 to RN	Coefficients of the previously occurring subcases (Real).

Remarks

1. A SUBSEQ command must only appear in a SUBCOM subcase.
2. A SUBSEQ command may be more than line of input. A comma at the end signifies a continuation.
3. SUBSEQ may only be used in statics or inertia relief problems.
4. A default value of 1.0 is used for all of the coefficients if no SUBSEQ command is used.

SUBTITLE - *Output Subtitle*

Description

Defines a BCD (character string) subtitle which will appear on the second heading line of each page of printer output.

Format and Examples

SUBTITLE = Any character data
SUBTITLE = NASTRAN PROBLEM NO. 5-1A

Remarks

1. SUBTITLE appearing at the subcase level will title output for that subcase only.
2. SUBTITLE appearing before all subcases will title any outputs which are not subcase dependent.
3. If no SUBTITLE is supplied, the subtitle line will be blank.
4. SUBTITLE information is also placed on NASTRAN plotter output as applicable.

SVECTOR - *Solution Set Displacement Output Request*

Description

Requests form and type of solution set displacement output.

Format and Examples

$$SVECTOR\left[\left(\left\{\begin{matrix} SORT1 \\ SORT2 \end{matrix}\right\}, \begin{matrix} \underline{PRINT} \\ NOPRINT, PUNCH, \\ PLOT \end{matrix} \begin{matrix} \underline{REAL} \\ IMAG \\ PHASE \end{matrix}\right)\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

SVECTOR = ALL

SVECTOR(SORT2, PUNCH, PHASE) = NONE

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number). SORT2 is available only in transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Displacements for all points (modes) will be output.
NONE	Displacements for no points (modes) will be output.
n	Set identification of previously appearing SET command. Only displacements of points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. In a frequency response problem any request for SORT2 causes all output to be SORT2.
4. SDISPLACEMENT is an alternate form and is entirely equivalent to SVECTOR.
5. SVECTOR = NONE allows overriding an overall output request.

SVELOCITY - *Solution Set Velocity Output Request*

Description

Requests form and type of solution set velocity output.

Format and Examples

$$SVELOCITY\left[\left(\left\{\begin{matrix} SORT1 \\ SORT2 \end{matrix}\right\}, \begin{matrix} \underline{PRINT} \\ NOPRINT, PUNCH \\ PLOT \end{matrix}, \left\{\begin{matrix} \underline{REAL} \\ IMAG \\ PHASE \end{matrix}\right\}\right)\right] = \left\{\begin{matrix} ALL \\ n \\ NONE \end{matrix}\right\}$$

SVELOCITY = 5

SVELOCITY(SORT2, PUNCH, PRINT, PHASE) = ALL

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number). SORT2 is available only in transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on frequency response problems.
ALL	Velocity for all solution points (modes) will be output.
NONE	Velocity for no solution points (modes) will be output.
n	Set identification of a previously appearing SET command. Only velocities of points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. Velocity output is only available for transient and frequency response problems.
4. In a frequency response problem any request for SORT2 output causes all output to be SORT2.
5. SVELOCITY = NONE allows overriding an overall output request.

SYM - *Symmetry Subcase Delimiter*

Description

Delimits and identifies a symmetry subcase.

Format and Examples

SYM n

SYM 123

Option	Meaning
n	Subcase identification number (Integer > 0).

Remarks

1. The subcase identification number, n, must be strictly increasing (that is, greater than all previous subcase identification numbers).
2. Plot requests and RANDOM requests should refer to n.
3. Overall output requests will not propagate into a SYM subcase (that is any output desired must be requested within the subcase).
4. SYM may only be used in statics or inertia relief.

SYMCOM - *Symmetry Combination Subcase Delimiter*

Description

Delimits and identifies a symmetry combination subcase.

Format and Examples

SYMCOM n

SYMCOM 123

Option	Meaning
n	Subcase identification number (Integer > 2).

Remarks

1. The subcase identification number, n, must be strictly increasing (that is, greater than all previous subcase identification numbers).
2. SYMCOM may only be used in statics or inertia relief problems.
3. Up to 360 SYMCOM commands can be used in one analysis.

SYMSEQ - *Symmetry Sequence Coefficients*

Description

Gives the coefficients for combining the symmetry subcases into the total structure.

Format and Examples

$SYMSEQ = R_1[,R_2, R_3, \ldots, R_n]$
SYMSEQ = 1.0, -2.0, 3.0, 4.0

Option	Meaning
R1 to RN	Coefficients of the previously occurring N SYM subcases (Real).

Remarks

1. A SYMSEQ command may only appear in a SYMCOM subcase.
2. The default value for the coefficients is 1.0 if no SYMSEQ command appears.
3. A SYMSEQ command may consist of more than one input line.
4. SYMSEQ may only be used in statics or inertia relief.

TEMPERATURE - *Thermal Properties Set Selection*

Description

Selects the temperature set to be used in either material property calculation or thermal loading.

Format and Examples

$$TEMPERATURE\left[\left(\left\{\begin{array}{c}MATERIAL\\LOAD\\\underline{BOTH}\\ESTIMATE\end{array}\right\}\right)\right] = n$$

TEMPERATURE (LOAD) = 15
TEMPERATURE (MATERIAL) = 7
TEMPERATURE = 7

Option	Meaning
BOTH	Both options, MATERIAL and LOAD, will use the same temperature table.
MATERIAL	The selected temperature table will be used to determine temperature-dependent material properties indicated on the MATTi type bulk data entries.
LOAD	The selected temperature table will be used to determine an equivalent static load.
n	Set identification number of TEMP, TEMPD, TEMPP1, TEMPP2, TEMPP3, TEMPRB, or TEMPAX bulk data entries (Integer > 0).

Remarks

1. Only one temperature-dependent material request may be made in any problem and it must be above the subcase level.
2. Thermal loading may only be used in statics, inertia relief, differential stiffness, and buckling problems.
3. Temperature-dependent materials may not be used in piece wise linear problems.
4. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement (SPC) loads.
5. Static, thermal, and element deformation loads must have unique set identification numbers.

TFL - *Transfer Function Set Selection*

Description

Selects the transfer function set to be added to the direct input matrices.

Format and Examples

TFL = n

TFL = 77

Option	Meaning
n	Set identification of a TF bulk data entry (Integer > 0).

Remarks

1. a TF bulk data entry will not be used unless selected by a TFL Case Control command.
2. Transfer functions are supported on dynamic solution sequences only.
3. Transfer functions are simply another form of direct matrix input wherein rows of the dynamic equation are specified.

TITLE - *Output Title*

Description

Defines a BCD (alphanumeric) title which will appear on the first heading line of each page of NASTRAN printer output.

Format and Examples

TITLE = Any character data

TITLE = **$// ABCDEFGHI .... $

Remarks

1. TITLE appearing at the subcase level will title output for that subcase only.
2. TITLE appearing before all subcases will title any outputs which are not subcase dependent.
3. If no TITLE is supplied, the title line will contain data and page numbers only.
4. TITLE information is also placed on NASTRAN plotter output as applicable.

TSTEP - *Transient Time Step Set Selection*

Description

Selects integration and output time steps for transient problems.

Format and Examples

TSTEP = n

TSTEP = 731

Option	Meaning
n	Set identification of a selected TSTEP bulk data entry (Integer > 0).

Remarks

1. A TSTEP bulk data entry must be selected by a TSTEP Case Control command.
2. Only one TSTEP bulk data entry may have this value of n.

VECTOR - *Displacement Output Request Description*

Requests form and type of displacement vector output.

Format and Examples

$$VECTOR\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}\right\}, \begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH\\PLOT\end{matrix}, \begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}\right)\right] = \left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

VECTOR = 5

VECTOR(REAL) = ALL

VECTOR(SORT2, PUNCH, REAL) = ALL

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available on transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point. SORT2 is available only in transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on complex eigenvalue or frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \leq$ phase < 360.0 degrees) on complex eigenvalue or frequency response problems.
ALL	Displacements for all points will be output.
NONE	Displacements for no points will be output.
n	Set identification of a previously appearing SET command. Only displacements of points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. On a frequency response problem any request for SORT2 causes all output to be SORT2.
3. DISPLACEMENT and PRESSURE are alternate forms and are entirely equivalent to VECTOR.
4. VECTOR = NONE allows overriding an overall output request.

VELOCITY - *Velocity Output Request*

Description

Requests form and type of velocity vector output.

Format and Examples

$$VELOCITY\left[\left(\left\{\begin{matrix}SORT1\\SORT2\end{matrix}\right\}, \begin{matrix}\underline{PRINT}\\NOPRINT,PUNCH,\\PLOT\end{matrix}, \begin{matrix}\underline{REAL}\\IMAG\\PHASE\end{matrix}\right)\right] = \left\{\begin{matrix}ALL\\n\\NONE\end{matrix}\right\}$$

VELOCITY = 5

VELOCITY(SORT2, PHASE, PUNCH) = ALL

Option	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence. SORT1 is not available in transient problems (where the default is SORT2).
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point. SORT2 is available only in transient and frequency response problems.
PRINT	The printer will be the output device.
PUNCH	The specifed output quantity will be written to a file having an extension, pch, appended to the name of the input file.
REAL or IMAG	Requests real or imaginary output on frequency response problems.
PHASE	Requests magnitude and phase ($0.0 \le$ phase < 360.0 degrees) on frequency response problems.
ALL	Velocity for all solution points will be output.
NONE	Velocity for no solution points will be output.
n	Set identification of a previously appearing SET command. Only velocities of points whose identification numbers appear on this SET command will be output (Integer > 0).

Remarks

1. Both PRINT and PUNCH may be requested.
2. An output request for ALL in transient and frequency response problems generally produces large amounts of printout. An alternative would be to define a SET of interest.
3. Velocity output is only available for transient and frequency response problems.
4. In a frequency response problem any request for SORT2 output causes all output to be SORT2.
5. VELOCITY = NONE allows overriding an overall output request.
6. SYSTEM cell 121 operates as a filter on the velocity output.by changing any terms with a magnitude less than $1.x10^{SYSTEM(121)}$ to 0.0 or (0.0,0.0) for complex values

Dollar Sign - $ - *Comment*

Description

Defines a comment by specifying a $ in column one with commentary text appearing in columns 2-80.

Format and Examples

$ any alphanumeric character string

$---THIS IS AN EXAMPLE OF A COMMENT.

Remarks

1. Unlike other Case Control commands, which are free field, the first character of a comment must have a $.

4 BULK DATA

The Bulk Data section defines the analysis model. Bulk Data statements define the model geometry, the connectivity between node points, constraints, and loads.

The data base for a NASTRAN-CORE run can be saved to a special file called the New Problem Tape (NPTP) using the CHKPNT Executive Control command. Using the restart file, the additional runs can be made without having to regenerate quantities such as the mass and stiffness matrix (The Advanced User might consider the use of AMSS as alternative).

For the initial run, called a cold start, the entire Bulk Data section must be submitted. Thereafter, if the original run was checkpointed, the Bulk Data file exists on the Problem Tape in sorted form where it may be modified and reused on restart. On restart, the bulk data contained in the current Bulk Data section are added to the bulk data contained on the Old Problem Tape (OPTP). The DELETE statement is used to remove bulk data statements from the OPTP.

Bulk data statements in the Bulk Data section can be submitted in any order since the Bulk Data input file is sorted before execution of the Input File Processor. It should be noted that the machine time to perform this is minimized for a file that is already sorted. The sort time for a badly sorted input file may be significant for large Bulk Data files.

The ECHO Case Control directive can be used to obtain a printout of either the unsorted or the sorted bulk data. A sorted echo is necessary in order to make modifications on a secondary execution using the Problem Tape. The default action is to print the sorted echo.

Format of Bulk Data Entries

Depending on type, each complete Bulk Data Entry may consist of either a single line of input or may span multiple lines. For historical reasons each line of input can contain, at most, 80 characters. Either of two formats may be used to specify an entry: Free field or fixed field. The fixed-field input format employs either 8-character or 16-character fields as described below. Free-field input format can only be used to generate 8-character fields where the fields are separated by commas or blanks. Free-field format offers the ability to automatically duplicate similar Bulk Data entries with minor changes in one or more selected fields.

Fixed-Field Input

The fixed-field input format is variable to the extent that any quantity except the mnemonic can be placed anywhere within a specified 8 or 16-character field. As noted above, each line of input contains 80 characters. For the small field data statement the 80 character entry is subdivided into 10 fields of 8 characters each.

Bulk Data Entries have a mnemonic tag in the first field. Data are then associated with fields 2-9. As each data statement is processed by the Input File Processor (IFP), the name of the entry is first compared with known entry names in the IFP table. If a match is found the data fields are checked for type and for required data fields and are then stored as data structures on one of several internal files as appropriate for the specific data type. If errors are found in this process a fatal error message is written on the output file and processing continues. If errors were found the program terminates after the Preface.

Small Field Bulk Data Entry

1	2	3	4	5	6	7	8	9	10

The data items are interpreted as characters by the IFP. No field can be more than 8 characters long. Each data field for a entry must be REAL, INTEGER or CHARACTER. A REAL must include a decimal point. An INTEGER must have no decimal point; and, a CHARACTER string must start with an ALPHA character. The datum in each field is read and typed by the IFP as REAL, INTEGER or CHARACTER. The IFP checks to make sure each datum is the proper type, that is, blank, integer, real, double precision, or character string

The presence of a decimal point defines a real number and the absence of a decimal point defines an integer number. All real numbers, including zero, must contain a decimal point. A blank may be interpreted as a real zero or integer zero. Generally speaking it is good practice to explicitly specify zeros since the default action is not consistent.

Real numbers may be encoded in various ways. For example, the real number 7.0 may be encoded as 7.0, .7E1, 0.7+1, 70.-1, .70+1, etc. A double precision number must contain both a decimal point and an exponent with the character D such as 7.0D0. Character data strings consist of one to eight alphanumeric characters, the first of which must be alphabetic.

Normally field 10 is reserved for optional user identification. However, in the case of continuations, field 10, starting with the 74th character is used in conjunction with field 1 of the a child statement as an identifier and must contain a unique entry. The continued data specification starts with a plus sign (+) followed by the same characters appearing starting with the 74th character in the parent. This allows the data to be submitted in unsorted form.

The small field data statement is adequate for data normally associated with structural engineering problems. Since abbreviated forms of floating point numbers are allowed, up to seven significant decimal digits may be used in an eight-character field. Occasionally, however, the input is generated by another computer program or is available in a form where a wider field would be desirable. For this case, the larger field format with a 16-character data field is provided. Each logical statement consists of two 80 character entries as indicated in the following diagram.

Large Field Bulk Data Entry

1	2	3	4	5	6

The large field statement is denoted by placing the star symbol (*) after the mnemonic in field 1a and some unique character string starting with character 74. The continuation entry starts with a * followed by the same character string used in the continuation field of the first entry. The continuation entry may in turn be used to point to a large or small field continuation entry, depending on whether a * or + appears in column 1. The use of multiple and large field entries is illustrated in the following examples.

Small Field statement with Small Field Continuation

1	2	3	4	5	6	7	8	9	10
PBAR	10	30	1.	10.	5.				+123
+123	.5	.5							

Large Field Statement

1	2	3	4	5	6
GRID*	10	0	1.	1.	*123
*123	1.	5	456		

In the above examples, character 73 arbitrarily contains the symbol + or * in all cases where field 10 is used as a pointer. However, character 73 could have been left blank or the same symbol used in character 1 of the following entry could have been used (that is, the symbols * or +).

Free-Field Input

The free-field input format can be used to create only small field statements. This capability is best understood by the following important rules and program features:

1. Free-field input is available only after BEGIN BULK and is disabled automatically when ENDDATA is read.
2. Free-field input is activated if the first eight characters contain one or more commas (,) or an equal sign (=).
3. Data items must be separated with a comma, one or more blanks, or the combination of a comma and blanks.
4. Integers and character fields are limited to 8 characters. Real numbers can be up to 12 characters, including sign and decimal point.
5. Duplication of fields from the preceding line of input is accomplished by using an equal sign (=) in the appropriate field.
6. Two equal signs (==) indicate duplication of all the trailing fields from the preceding line of input.
7. Incrementation of a value from the previous line of input is indicated by coding *(i), where i is the value of the increment (integer or floating point number) and * is the increment character. This feature is dependent on the corresponding field in the input statement. NOTE: The continuation fields on free field entries cannot be incremented automatically when using duplication.
8. Incrementation of a value from the previous line of input to an ending value is indicated by coding %(E), where E is the ending value (integer or floating point number) in the last line of input to be generated, and % is the ending character. This feature is also field dependent.
9. Repeated duplication is indicated by coding =(N), where N is the number of data entries to be generated using the value of the increment on the preceding (or current) entry by *(i), or the computed incremental value on the preceding entry by %(E).

10. A field index and value can be coded by n) X, where n is the field index and X the value.
11. The symbol)+ is equivalent to 10)+, where 10 is the tenth field of the input statement, which is normally the continuation ID field.
12. A right parenthesis) in the first character indicates the duplication of the tenth field of the preceding entry into the first field of the entry being generated.
13. The continuation ID (in field 1 or 10) is automatically increased (by 1) in the repeated-duplication operation. The ID must be in the form of +A-X, where A is one or more alphanumeric characters preceded by a plus and followed by a minus sign. X is an unsigned integer to be used as the initial value for the increment. A maximum of 8 characters (including signs) is allowed, with no embedded blanks. An “=(1)” in the first input field is needed for single entry duplication.
14. Data in field 10, not in the form of +A-X, is replaced by blanks during the repeated-duplication operation.
15. The ECHO Case Control command can be input (or redefined) at any time during the free-field input session.
16. Floating point numbers in the forms of 12300., 1.23E+04, or 1.23+4 are, acceptable. Twelve digits can be used for maximum accuracy. For example, 1234567890.1 is more accurate than 0.123456D+10.
17. The dollar sign ($) can be used freely as described elsewhere in this section.
18. Embedded blanks are not allowed in any double-character free-field input commands such as:

```
=(  *(    %(    )+    ==
```

19. Embedded blanks are not allowed in field 10, which is sometimes used as a comment field.
20. A slash (/), with or without a separator of comma or blank, indicates that the current field is the same as the previous field, for example

Free-Field Input Examples

The following examples illustrate the use of free-field input.

Example 1

```
GRID, 2, 3, 1.0 2.0,, 4,316
=, *(1), =, *(.2), == $
=(3)
```

The above free-field input will generate the following bulk data statements in NASTRAN 8-column field format:

1	2	3	4	5	6	7	8	9	10
GRID	2	3	1.0	2.0		4	316		
GRID	3	3	1.2	2.0		4	316		
GRID	4	3	1.4	2.0		4	316		
GRID	5	3	1.6	2.0		4	316		
GRID	6	3	1.8	2.0		4	316		

Example 2

```
grid,2,3,1.0,2.0,,4,316
=(4),*(1),=,%(1.8),==
```

The above entries will generate the same Bulk Data entries as in Example 1.

Example 3

```
Grid, 2 3 1.0 2.0, 7) 4, 316
```

This example will generate only one entry. This will be the same as the first entry in Example 1.

Example 4

```
Tabled3,62, 126.9, 30.0 10)+abc
),  1.23e+4,  5.67+8, 1234567. endt
```

This example will generate the following bulk data statements:

1	2	3	4	5	6	7	8	9	10
TABLED3	62	126.9	30.0						+abc
+abc	1.23+4	5.67+8	1234567.	ENDT					

Example 5

```
taBLed3, 62 126.9 30.0 )+aBc
```

This example will generate only one entry. This will be the same as the first statement in Example 4.

Example 6

```
This is only a test

THIS IS only a test

This, is only a test
```

The different results of the above three (3) input lines are shown by the following generated entries:

1	2	3	4	5	6	7	8	9	10
This is only a test									
THIS IS only a test									
THIS IS ONLY A TEST									

Example 7

```
PBAR, 3, 4, 5.0 , 6.0, )+ABC-1
= , *(1), =, *(2.)  ==
=(2)
+ABC-1, 7.7  8.8  9  )+DEF-22
=(3),==
```

This example will generate the following eight (8) bulk data with continuation ID fields automatically increased by 1.

1	2	3	4	5	6	7	8	9	10
PBAR	3	4	5.	6.					+ABC-1
PBAR	4	4	7.	6.					+ABC-2
PBAR	5	4	9.	6.					+ABC-3
PBAR	6	4	11.	6.					+ABC-4
+ABC-1	7.7	8.8	9						+DEF-1
+ABC-2	7.7	8.8	9						+DEF-2
+ABC-3	7.7	8.8	9						+DEF-3
+ABC-4	7.7	8.8	9						+DEF-4

Example 8

```
CQUAD4,    101    1     11   12  16  15
```

```
CQUAD4,     102     1       12    13   17   16
CQUAD4,     103     1       13    14   18   17
```

This example shows the combination of free-field and tabulation input. The requirement of 8 columns per field does not apply here.

Example 9

This example lists the input data using free-field Bulk Data entries used in NASTRAN Demonstration Problem No. D01-06-2A. It gives the same sorted input data as NASTRAN Demonstration Problem No. D01-06-1A, which uses the standard fixed-field bulk data.

```
ID D01062A,NASTRAN
APP     DISP
SOL     1,1
TIME    5
CEND
TITLE = SOLID DISC WITH RADIALLY VARYING THERMAL LOAD (FREE-FIELD)
SUBTITLE = NASTRAN DEMONSTRATION PROBLEM NO. D01-06-2A
LABEL = TRAPEZOIDAL RING ELEMENTS
ECHO  = BOTH
SPC   = 16
TEMPERATURE(LOAD) = 16
OUTPUT
SET 1 = 1,3,5,7,9,11,13,15,17,19,21,23,25,26
DISP  = 1
ELSTRESS =  ALL
BEGIN BULK
CTRAPRG, 1,1,3,4,2,.0,12
=(11), *(1) *(2),///, ==
GRDSET, 8)2456
GRID,1,,.0
=(3),*(2),,*(.005)
GRID,2,,.0,,.01
=(3),*(2),,*(.005),==
GRID,9,,.02
=(8),*(2),,%(.10)
GRID,10,,.02,,.01
=(8),*(2),,%(.10),==
MAT1,12,1.0+7,,.3,.2587-3,1.0-7,.0
SPC,16,1,13,.0,2,1,.0
TEMP,16,1,100.,2,100.,3,99.75
=,=,4,99.75,5,99.0,6,99.0
=,=,7,97.75,8,97.75,9,96.0
=,=,10,96.0,11,91.0,12,91.0
=,=,13,84.0,14,84.0,15,75.0
=,=,16,75.0,17,64.0,18,64.0
=,=,19,51.0,20,51.0,21,36.0
=,=,22,36.0,23,19.0,24,19.0
=,=,25,.0,26,.0
ENDDATA
```

Continuation Entries

Some Bulk Data entries require more than eight fields of data in the Small or Free Field formats, and the Large Field format requires (at least) two lines for each entry. Continuation entries are required in all such cases. A parent entry (the first line) is followed by one or more continuation entries on subsequent lines.

Continuation Indices

If an entry is continued, a unique pointer is entered into the final, 10^{th} field (6^{th} field for Large Field format) which matches the same identifier entered into the first field of the continuation entry.

Continuation indices are either entered explicitly by the user, or generated automatically by NASTRAN-CORE; automatic generation is the recommended practice. Automatic generation of continuation indices requires that a continuation entry immediately follows the entry for which it is a continuation.

Rules governing continuation entries and continuation indices are described in the next section.

Continuation Rules

1. The first field of a continuation entry must match the final field of its parent entry.
2. For automatic generation of continuation, the continuation line (or lines) must immediately follow the parent Bulk Data entry.
3. For automatic generation of the Free or Small Field formats, fields 1 and 10 of the continuation line (or lines) must be blank or a plus (+).
4. For automatic generation of the Large Field format, field 6 must be blank or an asterisk (*), but field 1 contains a single character, which must be an asterisk (*), and it must be in column 1 of the first field.
5. For automatic generation of continuation line, if a continuation line has no non-blank datum fields (fields two through eight for Small or Free Field formats, two through five in the case of the Large Field format). the field 1 must contain a plus (+) for Small Field Format or an asterisk (*) in column 1 for Large Field Format.

Continuation Index Rules

1. Any alphanumeric character may be used in the continuation indices. (However, see Note below).
2. The first character in the continuation index is not incremented.
3. The continuation fields are incremented by +1.
4. When generating continuation indices, NASTRAN-CORE uses indices of 7 digits starting with 1000001. The maximum number of continuation indices that can be generated is 8999998.

Examples

Free Field format examples:

Input – notice the example of the extraneous blank line between the two GRID entries which is ignored (correctly) by the bulk data sort – it is NOT a continuation. On the other hand, the PBAR entries could conceivably include an entirely blank continuation line, as shown here, and which is include (correctly) by the bulk data sort:

```
GRID,1,0,0.,0.

GRID,2,0,0.,180.
PBAR,1,1,1.,1.,1.,1.,1., ,2.5
SPC1,1,3,13,33,53,73,93
SPC1,1,12,1,2,3,4,5,6,21,22,23,24,25,26,41,42,43,44,45,61,62,63,64,81,82,83
ENDDATA
```

This results in the following Sorted Bulk Data Echo:

```
                S O R T E D   B U L K   D A T A   E C H O
    ---1--- +++2+++ ---3--- +++4+++ ---5--- +++6+++ ---7--- +++8+++ ---9--- +++10+++
1-      GRID    1       0       0.      0.
2-      GRID    2       0       0.      180.
3-      PBAR    1       1       1.      1.      1.      1.      1.              +1000001
4-      +1000001                                                                +1000002
5-      +10000022.5                                                             +1000003
6-      SPC1    1       3       13      33      53      73      93
7-      SPC1    1       12      1       2       3       4       5       6       +1000004
8-      +100000421      22      23      24      25      26      41      42      +1000005
9-      +100000543      44      45      61      62      63      64      81      +1000006
10-     +100000682
ENDDATA
```

Small Field Examples:

```
I N P U T   B U L K   D A T A   D E C K   E C H O

    ---1--- +++2+++ ---3--- +++4+++ ---5--- +++6+++ ---7--- +++8+++ ---9--- +++10+++
      GRID    92              10.     80.     .82
      GRID    93              30.     80.     .82
      MAT1    1       10.4+6  4.+6
      SPC1    1       1       11      31      51      71      91
      SPC1    1       3       13      33      53      73      93
      SPC1    1       12      1       2       3       4       5       6
             21      22      23      24      25      26      41      42
             43      44      45      61      62      63      64      81
             82      83
      ENDDATA

S O R T E D   B U L K   D A T A   E C H O
    ---1--- +++2+++ ---3--- +++4+++ ---5--- +++6+++ ---7--- +++8+++ ---9--- +++10+++
1-      GRID    92              10.     80.     .82
2-      GRID    93              30.     80.     .82
3-      MAT1    1       10.4+6  4.+6
4-      SPC1    1       1       11      31      51      71      91
5-      SPC1    1       3       13      33      53      73      93
6-      SPC1    1       12      1       2       3       4       5       6       +1000001
7-      +100000121      22      23      24      25      26      41      42      +1000002
8-      +100000243      44      45      61      62      63      64      81      +1000003
9-      +100000382      83                                                      +1000004
ENDDATA
```

Large Field Example:

```
I N P U T   B U L K   D A T A   D E C K   E C H O

 ---1--- +++2+++ ---3--- +++4+++ ---5--- +++6+++ ---7--- +++8+++ ---9--- +++10+++
GRID*   92                              10.             80.
        .82
GRID    93              30.     80.     .82
MAT1    1       10.4+6  4.+6
PBAR*   1               1               1.              1.
        1.              1.              1.

                                                        2.5
```

```
SPC1    1       1       11      31      51      71      91
SPC1    1       3       13      33      53      73      93
SPC1    1       12      1       2       3       4       5       6
        21      22      23      24      25      26      41      42
        43      44      45      61      62      63      64      81
        82      83
ENDDATA

S O R T E D   B U L K   D A T A   E C H O
 ---1--- +++2+++ ---3--- +++4+++ ---5--- +++6+++ ---7--- +++8+++ ---9--- +++10+++
GRID*   92                              10.             80.             *1000001
*1000001.82
GRID    93              30.     80.     .82
MAT1    1       10.4+6  4.+6
PBAR*   1               1               1.              1.              *1000002
*10000021.              1.              1.                              *1000003
*1000003                                                                *1000004
*1000004                                                2.5
SPC1    1       1       11      31      51      71      91
SPC1    1       3       13      33      53      73      93
SPC1    1       12      1       2       3       4       5       6       +1000005
+100000521      22      23      24      25      26      41      42      +1000006
+100000643      44      45      61      62      63      64      81      +1000007
+100000782      83
ENDDATA
```

Bulk Data Entry Descriptions

Detailed descriptions of the Bulk Data Entries are contained in this section in alphabetical order. Small field examples are given for each statement along with a description of the contents of each field. In the Format and Example sections of each entry description, both a symbolic data statement format description and an example of an actual statement are shown. Literal constants are shown in the format section enclosed in quotes (for example, “0” and “THRU”.) Fields that are required to be blank are indicated in the format section by a blank box.

The Input File Processor will produce error messages for any entries that do not have the proper format or that contain illegal data.

Continuations need not be present unless they contain required data. In the case of multiple continuation entries, the intermediate lines of input must be present (even though fields 2-9 are blank), if one of the following lines contains data in fields 2-9. In addition, a double field format requires at least two input lines (or subsequent multiples of two) so that 10 data fields are included, even if this leads to one or more double field entries that do not contain any data.

Comment

Description

Comments may appear anywhere within the input file and are ignored by the program. Further, they will not be present in a sorted echo of the Bulk Data section, or on the database.

Format

1	2	3	4	5	6	7	8	9	10
$									

Example

$ This is a comment that can be inserted anywhere in the input file

Remarks

1. Comments appear only in the UNSORTED echo of the Bulk Data file.

Delete

Description

Removes entries on restart or from the User’s Master File.

Format

1	2	3	4	5	6	7	8	9	10
/	K1	K2							

Example

/	4	85							

Field	Description
K1	Sorted sequence number of first data entry in sequence to be removed. (Integer > 0)
K2	Sorted sequence number of last data entry in sequence to be removed. (Integer > 0; Default = K1)

Remarks

1. The DELETE entry removes Bulk Data entries having sort sequence numbers K1 through K2, inclusive.
2. If K2 is blank, only entry K1 is removed from the Bulk Data Section.
3. DELETE entries are ignored in non-restart runs.

ADUMi - *User-Defined Element Attributes*

Description

Defines attributes of a user-defined (dummy) element and its associated entries (1≤ i ≤ 9).

Format

1	2	3	4	5	6	7	8	9	10
ADUMi	NG	NC	NP	ND					

Example

ADUM1	8	2	1	3					

Field	Description
NG	Number of grid points connected by element (Gi fields on CDUMi.) (Integer > 0)
NC	Number of additional fields (Ai) on CDUMi connection entry. (Integer ≥ 0)
NP	Number of additional fields (Ai) on PDUMi property entry. (Integer ≥ 0)
ND	If nonzero (blank or 0 implies no differential stiffness), number of displacement components at each grid point to be used in generation of differential stiffness matrix. (Blank or integer 0, 3, or 6)

ASET - *Analysis Set (a-set) Definition*

Description

Defines degrees of freedom to be placed in the analysis set (*a*-set).

Format

1	2	3	4	5	6	7	8	9	10
ASET	ID	C	ID	C	ID	C	ID	C	

Example

ASET	16	2	23	3516			1	4	

Field	Description
ID	Grid or scalar point identification numbers.
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. Degrees of freedom selected on this entry are placed into the mutually exclusive *a*-set, and may not be referenced on entries that define other mutually-exclusive sets.
2. When ASET or ASET1 entries are present, all degrees of freedom not otherwise constrained (e.g. on SPC or MPC-type entries) or referenced on a SUPORT entry will be placed in the omitted degree of freedom set (*o*-set).

ASET1 - *Analysis Set (a-set) Definition, Alternate Form*

Description

Defines degrees of freedom to be placed in the analysis set (*a*-set). Alternate form of the ASET entry.

Format

1	2	3	4	5	6	7	8	9	10
ASET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example

ASET1	345	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example

ASET1	C	ID1	“THRU”	ID2					
ASET1	123456	7	THRU	109					

Field	Description
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0)

Remarks

1. Degrees of freedom selected on this entry are placed into the mutually exclusive *a*-set, and may not be referenced on entries that define other mutually-exclusive sets.
2. When ASET or ASET1 entries are present, all degrees of freedom not otherwise constrained (e.g., on SPC or MPC-type entries) or referenced on a SUPORT entry will be placed in the omitted degree of freedom set (*o*-set).
3. If this alternate form is used, all of the grid or scalar points ID1 through ID2 must be defined.

AXIC - *Axisymmetric Problem Flag*

Description

Defines the existence of a model containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
AXIC	H								

Example

AXIC	15								

Field	Description
H	Highest harmonic index defined in the problem. (Integer; 0 < H < 998)

Remarks

1. Only one AXIC entry is allowed and, when present, disallows most other Bulk Data entries. Those types which are allowed are listed below.

CCONEAX	MAT1	SECTAX
CTRAPAX	MATT1	SPCADD
CTRIAAX	MOMAX	SPCAX
DAREA	MOMENT	SUPAX
DELAY	MOMENT	TABDMP1
DLOAD	MPCADD	TABLED1
DMI	MPCAX	TABLED2
DMIG	NOLIN1	TABLED3
DPHASE	NOLIN2	TABLED4
EIGB	NOLIN3	TABLEM1
EIGC	NOLIN4	TABLEM2
EIGR	OMITAX	TABLEM3
EPOINT	PARAM	TABLEM4
FORCE	PCONEAX	TEMPAX
FORCEAX	POINTAX	TF
FREQ	PRESAX	TIC
FREQ1	RINGAX	TLOAD1
FREQ2	RFORCE	TLOAD2
GRAV	RLOAD1	TSTEP
LOAD	RLOAD2	

2. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
3. For a discussion of the axisymmetric solid problem, see Section 5.11 of the Theoretical Manual.

BAROR - *CBAR Entry Default Values*

Description

Provides default values for field 3 and fields 6 through 8 of the CBAR entry.

Format

1	2	3	4	5	6	7	8	9	10
BAROR		PID			X1	X2	X3		

Example

BAROR		39			0.6	2.9	-5.87		

Alternate Format and Example

BAROR		PID			G0				
BAROR		45			20				

Field	Description
PID	PBAR entry property identification number. (Integer > 0 or blank)
X1,X2,X3	Components of an orientation vector, $\vec{V}$, from GA, in the displacement coordinate system at GA. See Remark 4. (Real or blank)
G0	Grid number of a point used to define the element orientation vector, $\vec{V}$. The vector direction is from GA to G0. See Remark 5. (Integer > 0, G0 ≠ GA, GB on CBAR entry)

Remarks

1. The BAROR entry provides default values for corresponding blank CBAR entry fields.
2. Only one BAROR is allowed.
3. For an explanation of bar element geometry, see **CBAR** (p. 233).
4. If field 6 is a real value (contains a decimal point), then fields 7 and 8 must likewise contain real values. Together they specify the X1, X2 and X3 orientation vector components.
5. If field 6 is an integer then the alternate format is assumed. In this case fields 7 and 8 must be blank.

BDYC - *Combination of Substructure Boundary Sets*

Description

Defines a boundary set combination from a list of substructures and substructure boundary sets. The grids and components in the combined boundary set may then be used in subsequent CREDUCE, MREDUCE, or REDUCE operations.

Format

1	2	3	4	5	6	7	8	9	10
BDYC	ID	NAME1	SID1	NAME2	SID2	NAME3	SID3		
		NAMEi	SIDi	-etc.-					

Example

BDYC	157	WINGRT	7	MIDMG	15	FUSELAG	32		
		POD	175	WINGRT	15	CABIN	16		

Field	Description
ID	Identification number of combination boundary set. (Integer > 0)
NAMEi	Name of basic substructure which contains the grid points defined by boundary set SIDi. (Character)
SIDi	Identification number of the boundary set associated with basic substructure NAMEi. (Integer > 0)

Remarks

1. Boundary sets must be selected in the Substructure Control Section (BOUNDARY = ID) to be used by NASTRAN-CORE. Note that “BOUNDARY” is a subcommand of the substructure CREDUCE, MREDUCE, and REDUCE commands.
2. The same substructure name may appear more than once on a BDYC entry.
3. The SIDi boundary set IDs are defined on the BDYS and BDYS1 entries and need not be unique.
4. BDYC ID fields must be unique across all BDYC entries.
5. After two or more basic substructures are combined, the connected degrees of freedom are actually the same and may be referenced with any one of the substructure names. Redundant specification is allowed.

BDYS - *Substructure Boundary Set Definition*

Description

Defines basic substructure boundary degrees of freedom. The boundary set is used in the substructure REDUCE, CREDUCE, and MREDUCE operations.

Format

1	2	3	4	5	6	7	8	9	10
BDYS	SID	G1	C1	G2	C2	G3	C3		

Example

BDYS	7	13	123456	15	123	17	123456		

Field	Description
SID	Identification number of BDYS set. (Integer > 0)
Gi	Identification number of a grid or scalar point in the basic substructure. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. The set of boundary points defines the degrees of freedom which are to be retained in the matrices after the substructure REDUCE, CREDUCE or MREDUCE operation has been performed. The BDYS1 entry provides an alternate format.
2. The SID need not be unique.
3. A BDYS entry must be referenced by a BDYC entry in order to associate a basic substructure name with a boundary set specified on the BDYS. Note that the same BDYS boundary set may be attached to more than one basic substructure name.

BDYS1 - *Substructure Boundary Set Definition, Alternate Form*

Description

Defines basic substructure boundary degrees of freedom. The boundary set is used in the substructure REDUCE, CREDUCE, and MREDUCE operations.

Format

1	2	3	4	5	6	7	8	9	10
BDYS1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

Example

BDYS1	15	123456	275	276	277	457	589	102	
	103	105		1275					

Alternate Format and Example

BDYS1	SID	C	G1	G2	-etc.-	Gj	“THRU”	Gk	
BDYS1	25	1236	105	THRU	305	450	451		

Field	Description
SID	Identification number of BDYS1 set. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Gi	Identification number of a grid or scalar point in the basic substructure. (Integer > 0)

Remarks

1. The set of boundary points defines the degrees of freedom which are to be retained in the matrices after the substructure REDUCE, CREDUCE or MREDUCE operation has been performed. The BDYS entry provides an alternate format.
2. Though shown as an alternate format, the “THRU” option is actually supported at any point within the grid/scalar point list, though it may only appear in fields 3 through 8.
3. The SID need not be unique.
4. A BDYS1 entry must be referenced by a BDYC entry in order to associate a basic substructure name with a boundary set specified on the BDYS1. Note that the same BDYS1 boundary set may be attached to more than one basic substructure name.

CBAR - *Simple Beam Element Connection*

Description

Defines a simple beam element.

Format

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Example

CBAR	2	39	7	3	1.5	1.5	.2		
		513							

Alternate Format

CBAR	EID	PID	GA	GB	G0				
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Example

CBAR	2	39	7	3	13				
		513							

Field	Description
EID	Element identification number. (Integer > 0)
PID	.Identification number of a PBAR property entry (Default=EID unless BAROR entry has nonzero entry in field 3). See BAROR entry for default options. (Integer > 0 or blank)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{V}$, in Plane 1 with origin at End A, measured in displacement coordinate system at GA. Defaults may be provided on the BAROR entry. (See **Figure 4-8**.) (Real; $X1^2 + X2^2 + X3^2 > 0.0$ or blank)
G0	Grid point identification number to optionally supply X1, X2, X3. Default value may be provided on the BAROR entry. (Integer > 0 or blank)
PA, PB	Pin flags for bar ends A and B, respectively. Components listed in the pin flags will be released from the element, and it will not be able to resist motion in the corresponding force or moment degrees of freedom. Note: The element must have stuffiness associated with the degrees of freedom in the pin flag. For example, if a pin flag of “4” is given, the CBAR must have a value for the torsional constant, J. (Integer > 0 or Blank; Up to 5 of the unique digits 1 through 6 with no embedded blanks.)
W1A,W2A,W3A; W1B,W2B,W3B	Components of offset vectors wa and wb, respectively, (see **Figure 4-8**) in displacement coordinate systems at points GA and GB. (Real or blank)

Remarks

1. CBAR element identification numbers must be unique with respect to all other element identification numbers.
2. The continuation entry is optional.

3. If bar offset vectors are present, NASTRAN-CORE plotting will plot the bar connecting to the tip of the offset, not to the associated grid point.

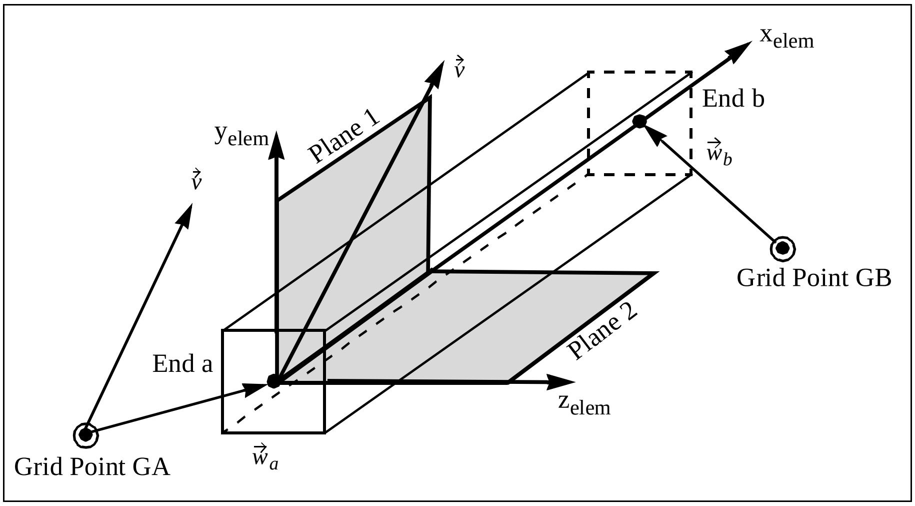


Figure 4-1 CBAR Element Coordinate System and Geometry

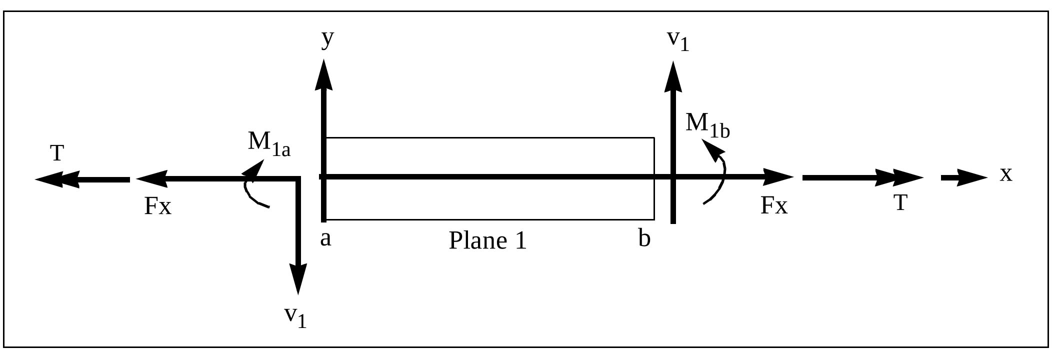


Figure 4-2 CBAR Forces and Moments, Plane 1

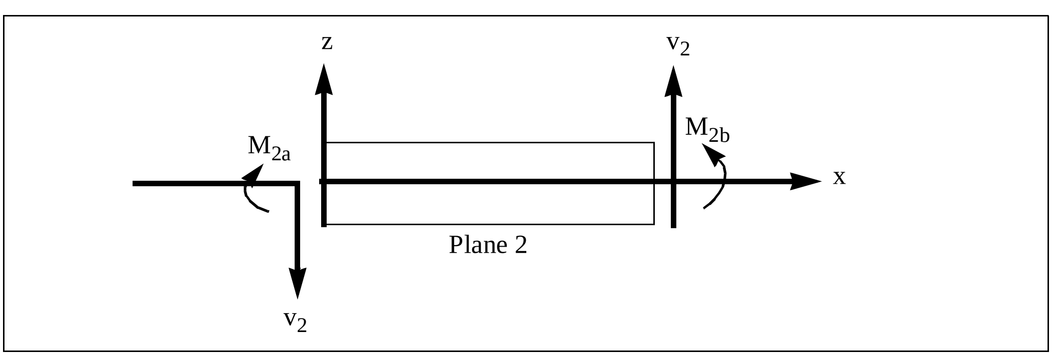


Figure 4-3 CBAR Forces and Moments, Plane 2

CBARAO - *Select intermediate locations for stress and force output on BAR elements*

Description

Selects additional locations on BAR elements for stress and force output.

Format

1	2	3	4	5	6	7	8	9	10
CBARAO	EID	SCALE	X1	X2	X3	X4	X5	X6	

Example

CBARAO	24	LE	1.2	2.4	3.6	4.8			

Alternate Format

CBARAO	EID	SCALE	NPTS	XS	ΔX				

Example

CBARAO	24	LE	4	1.2	1.2				

Field	Description
EID	Element identification number. (Integer > 0)
SCALE	.Method used to define X_i values (Character - either “LE” or “FR”)
X1, X2, X3...	Locations along the element where stress and force output is desired. If SCALE = LE, then they are distances from the start of the element. If SCALE = FR, then they are ratios of the distance to the length of the element.
NPTS	Number of locations to be defined
XS	Starting location for stress and force output (Used for X1 in remark 2). Distance defined in a similar manner to X_i.
ΔX	Incremental distance along element axis between points (distances defined in a similar manner to X_i)

Remarks

1. By default, stress adn force output is only available at the start and end of a BAR element. If entry is provided for a BAR (and if a PLOAD1 is used to define a load on the bar), stress and force output will be calculated at intermediate locations in addition to the end points.
2. When the alternate format is used, $X_i = X_{i-1} + DX$ (where i = 2,3,...NPTS)
3. Stress and force output generated using this command will include the effect of loads defined using PLOAD1 entries. Note - if PLOAD1 entries are not used, the forces are constant and the moments are linear along the length and intermediate output should not be necessary.
4. A maximum of 19 intermediate locations may be defined a value of FR for SCALE, a maximum of 6 using LE for SCALE..

CBEAM - *Alternate Simple Beam Element Connection*

Description

Defines a simple beam element.

Format

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	X1	X2	X3	CSys	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

Example

CBEAM	2	39	7	3	1.5	1.5	.2		
		513							

Alternate Format

CBEAM	EID	PID	GA	GB	G0			ABG	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
SA	SB								

Example

CBEAM	2	39	7	3	13				
		513							

Field	Description
EID	Element identification number. (Integer > 0)
PID	.Identification number of a PBEAM property entry. (Integer > 0 or blank)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{V}$ as shown in **Figure 4-4** , in Plane 1 with origin at End A, measured in the coordinate system selected by ABG. (Real; $X1^2 + X2^2 + X3^2 > 0.0$ or blank)
G0	Grid point identification number to optionally supply X1, X2, X3. Default value may be provided on the BAROR entry. (Integer > 0 or blank)
CSys	Alphanumedric code to specify the coordinate systems used to define the offsets at ends A and B and also the orientation vector. If used, it must be 3 characters long. The default is GGG (or the Global system for all). Acceptable characters are: E - Element coordinate system B - Basic coordinate system G - Global (or displacement) coordinate system If the field is non-bland, then it must contain 3 characters in the following order 1st character (E, B, or G) coordinate system used to define the offset at end A 2nd character (E, B, or G) - coordinate system used to define the offset at end B 3rd character (B or G) - coordinate system used to define orientation vector $\vec{V}$

Field	**Description**
PA, PB	Pin flags for bar ends A and B, respectively. Components listed in the pin flags will be released from the element, and it will not be able to resist motion in the corresponding force or moment degrees of freedom. Note: The element must have stuffiness associated with the degrees of freedom in the pin flag. For example, if a pin flag of "4" is given, the PBEAM must have a value for the torsional constant, J. (Integer > 0 or Blank; Up to 5 of the unique digits 1 through 6 with no embedded blanks.)
W1A,W2A,W3A; W1B,W2B,W3B	Components of offset vectors wa and wb, respectively, (see **Figure 4-8**) in displacement coordinate systems at points GA and GB. (Real or blank)
SA, SB	Ignored in current implementation (see remark 4.) Blank or integer.

Remarks

1. CBEAM element identification numbers must be unique with respect to all other element identification numbers.
2. An integer value in field 6 indicates that G0 is being used, a real value indicates that you are using X1, X2, and X3 to define the orientation vector. If G0 is used, then it cannot lie on the line connecting the start and end of the element.
3. The continuation entries are optional. However, if the second continuation entry is used, then the first one must be present, even if all fields on it are blank.
4. The current implementation does not support warping dof.
5. If offset vectors are present, NASTRAN-CORE plotting will plot the beam connecting to the tip of the offset, not to the associated grid point.

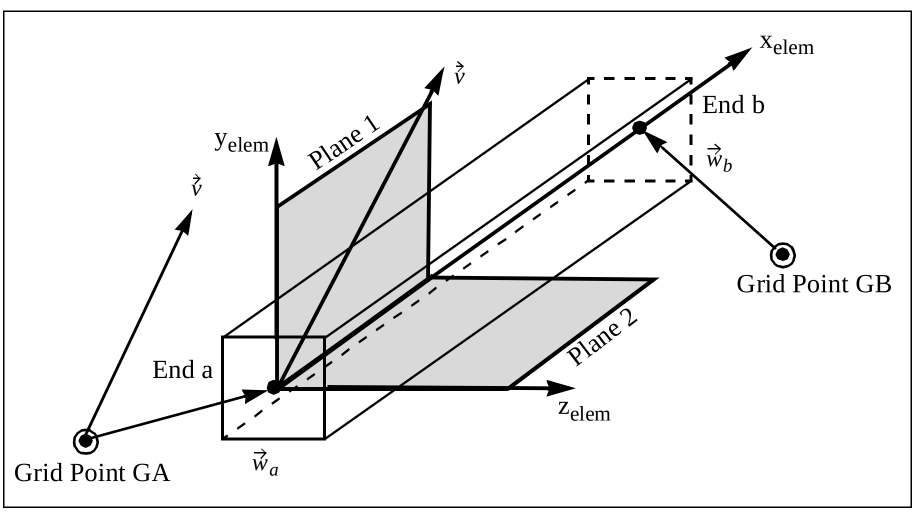


Figure 4-4 CBEAM Element Coordinate System and Geometry

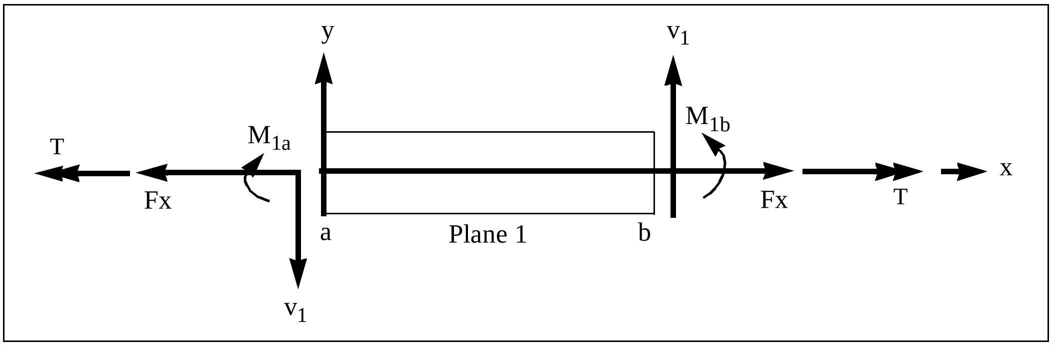


Figure 4-5 CBEAM Forces and Moments, Plane 1

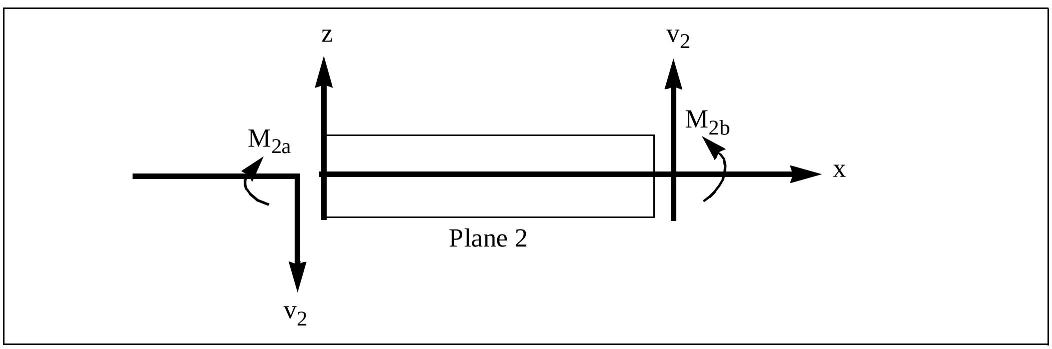


Figure 4-6 CBEAM Forces and Moments, Plane 2

CBUSH - *General Spring/Damper/Mass connectivity*

Description

Defines a spring/mass/damper between two grid points

Format

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

Example using the orientation vector

CBUSH	2	39	7	3	1.5	1.5	.2		
		513	1.	2.6	0.3				

Example using a reference GRID point

CBUSH	2	39	1010					3	

Field	Description
EID	Element identification number. (Integer > 0)
PID	.Identification number of a PBUSH property entry (Integer > 0 or blank)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{V}$, in Plane 1 with origin at End A, measured in displacement coordinate system at GA. (Real or blank)
G0	Grid point identification number to optionally supply X1, X2, X3. (Integer > 0 or blank)
CID	Element coordinate system. 0 means the BASIC system, a blank field means use $\vec{V}$ to determine the element coordinate system. (Integer or blank)
S	Fractional location of spring/damper/mass on the line connecting grids GA and GB (Real, default = 0.5)
OCID	Coordinate system used to define the element centroid. (Integer or blank)
S1,S2,S3	Offset vector (In OCID) for element from GA if OCID is provided (Real or blank)

Remarks

1. CBUSH element identification numbers must be unique with respect to all other element identification numbers.
2. The continuation entry is optional. If both GA and GB are provided, the element centroid defaults to half way between the two grid points. If only GA is provided, it defaults to at GA.
3. Springs and Dampers are handled in a manner similar to the following. If you were to place coincident points at the element centroid and connect them with the springs and dampers. Then if you were to connect one of the coincident points to GA using an RBAR element and the other coincident point to GB using an RBAR element.
4. Mass is handled as a luimped mass (similar to a CONM) located at the element centroid and the mass is transferred to GA and GB in a manner which retains the rigid-body properties of the mass..
5. If GA and GB are coincident or if GB is not present, then CID must be used to define the element coordinate system.

6. The orientation of the element must be defined. It may be defined using one of the three available methods. If CID is provided, then X1, X2, X3, and G0 are ignored.

Method	Results
X1,X2,X3	The element coordinate system is defined in a manner similar to a BAR element using X1, X2, and X3. This method may only be used if both GA and GB are provided and they are not coincident.
G0	The element coordinate system is determined in a manner similar to a BAR element using G0 This method may only be used if both GA and GB are provided and they are not coincident.
CID	The element coordinate system is defined by CID (calculated at the element centroid if CID is cylindrical or spherical)

7. The location of the centroid of the element is defined using the continuation entry. If OCID is provided, then S1, S2, and S3 are used to determine the centroid (offset from GA). Otherwise, S will be used to determine the centroid.
8. The BUSH element is preferred to using CELASi adn CDAMPi entries, as the element automatically accounts for offsets and displacement coordinate systems and prevents unintentional grounding. (NOTE: if only GA is specified, then you are defining spring/dampers to ground)

CCONEAX - *Axisymmetric Shell Element Connection*

Description

Defines a conical shell element.

Format

1	2	3	4	5	6	7	8	9	10
CCONEAX	EID	PID	RA	RB					

Example

CCONEAX	1	2	3	4					

Field	Description
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PCONEAX entry (Integer > 0; Default = EID).
RA	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)
RB	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)

Remarks

1. CCONEAX entries are allowed only if an AXIC entry is also present.

CDAMP1 - *Scalar Damper Connection*

Description

Defines a scalar damper element.

Format

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

Example

CDAMP1	19	219	6		6	2			

Field	Description
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PDAMP property entry (Integer > 0 or blank; Default = EID)
G1, G2	Grid point identification number. (Integer ≥ 0 or blank)
C1, C2	Degree of freedom component number. (0 ≤ Integer ≤ 6 or blank)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded connection G1 or G2 with a corresponding blank or zero C1 or C2. If only scalar points and/or ground are involved, it is more efficient to use CDAMP3. (A grounded connection is a scalar point or grid point degree of freedom whose displacement is constrained to zero.)
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. The two connection points, (G1, C1) and (G2, C2), must be distinct..

CDAMP2 - *Scalar Damper Properties and Connection*

Description

Defines a scalar damper element without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

Example

CDAMP2	19	2.98	6		6	2			

Field	Description
EID	Element identification number. (Integer > 0)
B	Damping coefficient. (Real)
G1, G2	Grid point identification number. (Integer ≥ 0 or blank)
C1, C2	Component number. (0 ≤ Integer ≤ 6 or blank)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded connection G1 or G2 with a corresponding blank or zero C1 or C2. If only scalar points and/or ground are involved, it is more efficient to use CDAMP3. (A grounded connection is a scalar point or grid point degree of freedom whose displacement is constrained to zero.)
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. The two connection points, (G1, C1) and (G2, C2), must be distinct.

CDAMP3 - *Scalar Damper Connection to Scalar Points*

Description

Defines a scalar damper element connected to scalar points only.

Format

1	2	3	4	5	6	7	8	9	10
CDAMP3	EID	PID	S1	S2					

Example

CDAMP3	19	698	6	10					

Field	Description
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PDAMP property entry (Integer > 0 or blank; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0 or blank; S1 ≠ S2)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A zero or blank S1 or S2 indicates a grounded scalar point.
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.

CDAMP4 - *Scalar Damper Properties and Connection to Scalar Points*

Description

Defines a scalar damper element connected to scalar points only and without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CDAMP4	EID	B	S1	S2					

Example

CDAMP4	19	2.98	6	10					

Field	Description
EID	Element identification number. (Integer > 0)
B	Damping value. (Real)
S1, S2	.Scalar point identification numbers. (Integer ≥ 0 or blank; S1 ≠ S2)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A zero or blank S1 or S2 indicates a grounded scalar point
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.

CDUMi - *User-Defined Element Connection*

Description

Defines a user-defined (dummy) element (1 ≤ i ≤ 9).

Format

1	2	3	4	5	6	7	8	9	10
CDUMi	EID	PID	G1	G2	G3	G4	-etc.-	GN	
	A1	A2	-etc.-	AN					

Example

CDUM2	114	108	2	5	6	8	11		
	2.4		3.4	2	50				

Field	Description
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PDUMi property entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 through GN must be unique).
Ai	Additional fields. (Real or integer)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The associated element routines for matrix generation, stress recovery, etc., must be coded and linked to replace the dummy routines.
3. If a PDUMi property entry is not needed, PID (field 3) may contain the material identification number.
4. Additional fields are defined in the user-written element routines.

CELAS1 - *Scalar Spring Connection*

Description

Defines a scalar spring element.

Format

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

Example

CELAS1	19	109	6	2	25	2			

Field	Description
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PELAS property entry. (Integer > 0 or blank; Default = EID)
G1, G2	Grid point identification number. (Integer ≥ 0 or blank)
C1, C2	Component number. (0 ≤ Integer ≤ 6 or blank)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded connection G1 or G2 with a corresponding blank or zero C1 or C2. If only scalar points and/or ground are involved, it is more efficient to use CELAS3. (A grounded connection is a scalar point or grid point degree of freedom whose displacement is constrained to zero.)
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. The two connection points, (G1, C1) and (G2, C2), must be distinct.

CELAS2 - *Scalar Spring Properties and Connection*

Description

Defines a scalar spring element without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	Ge	S	

Example

CELAS2	19	1.E5	6	2	25	2			

Field	Description
EID	Element identification number. (Integer > 0)
K	Spring stiffness. (Real)
G1, G2	Grid point identification number. (Integer ≥ 0 or blank)
C1, C2	Component number. (0 ≤ Integer ≤ 6 or blank)
Ge	Structural damping coefficient (Real, devault = 0.0)
S	Stress coefficient (Real, default = 0.0)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded connection G1 or G2 with a corresponding blank or zero C1 or C2. If only scalar points and/or ground are involved, it is more efficient to use CELAS3. (A grounded connection is a scalar point or grid point degree of freedom whose displacement is constrained to zero.)
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. The two connection points, (G1, C1) and (G2, C2), must be distinct.

CELAS3 - *Scalar Spring Connection to Scalar Points*

Description

Defines a scalar spring connected to scalar points only.

Format

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

Example

CELAS3	19	698	6	10					

Field	Description
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PELAS property entry. (Integer > 0 or blank; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0 or blank; S1 ≠ S2)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.

CELAS4 - *Scalar Spring Properties and Connection to Scalar Points*

Description

Defines a scalar spring element connected to scalar points only and without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

Example

CELAS4	19	1.E5	6	10					

Field	Description
EID	Element identification number. (Integer > 0)
K	Spring stiffness. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0 or blank; S1 ≠ S2)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.

CGAP - *Definition of a GAP*

Description

Defines a GAP constraint.

Format

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	X1	X2	X3	CID	

Example

CGAP	2	39	7	3	1.5	1.5	.2		

Alternate Format

CGAP	EID	PID	GA	GB	GO			CID	

Example

CGAP	2	39	1010	1000	10			3	

Field	Description
EID	Element identification number. (Integer > 0)
PID	.Identification number of a PGAP property entry (Integer > 0 or blank)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{V}$, in Plane 1 with origin at End A, measured in displacement coordinate system at GA. (Real or blank)
G0	Grid point identification number to optionally supply X1, X2, X3. (Integer > 0 or blank)
CID	Element coordinate system. 0 means the BASIC system, a blank field means use $\vec{V}$ to determine the element coordinate system. (Integer or blank)

Remarks

1. CGAP entries and their associated PGAP entries are internally converted in the program into GAP entries. This generates a constraint equation, not an element.
2. The generated GAP entries have no stiffness associated with them. They are simply used to measure a GAP (exception = SOL’s 1 and 2, which support the iterative GAP constraint).
3. Fields X1, X2, X3, and GO are not used in the conversion. Only GA and GB are used from the CGAP entry.
4. The initial opening is defined on the PGAP entry, not by the distance between GA and GB (If you use a GAP entry, the initial opening defaults to the distance between GA and GB).
5. See the GAP entry for a definition of the GAP constraint.

CELBOW - *Curved Beam Connection*

Description

Defines a curved beam element.

Format

1	2	3	4	5	6	7	8	9	10
CELBOW	EID	PID	GA	GB	X1	X2	X3	GEOM	

Example

CELBOW	29	2	3	45	-1.0	0.0	0.0	1	

Field	Description
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PELBOW property entry. (Integer > 0)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA $\neq$ GB)
X1, X2, X3	Components of orientation vector $\vec{V}$, in Plane 1 with origin at End A, measured in displacement coordinate system at GA. $\vec{V}$ points in the direction of C (center of curvature). (Real; $X1^2 + X2^2 + X3^2 > 0.0$; Default Xi = 0.0) (See **Figure 4-11**.)
GEOM	Element connection option flag. See Remark 6. (Integer = 1)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Pin flags or offsets are not supported by the CELBOW element.
3. The product moment of inertia is neglected (I12 = 0.0). Note that this assumes at least one axis of cross-sectional symmetry, for example, tube, I-beam, channel, tee, etc.
4. The local element coordinate system is shown in **Figure 4-11**. Plane 1 contains the points GA and GB and the orientation vector $\vec{V}$. Plane 2 is defined as being normal to Plane 1.
5. Element forces and stresses are expressed in the element coordinate system at End A, and in a rotated tangent coordinate system at End B.
6. GEOM = 1 is the only option available at this time.

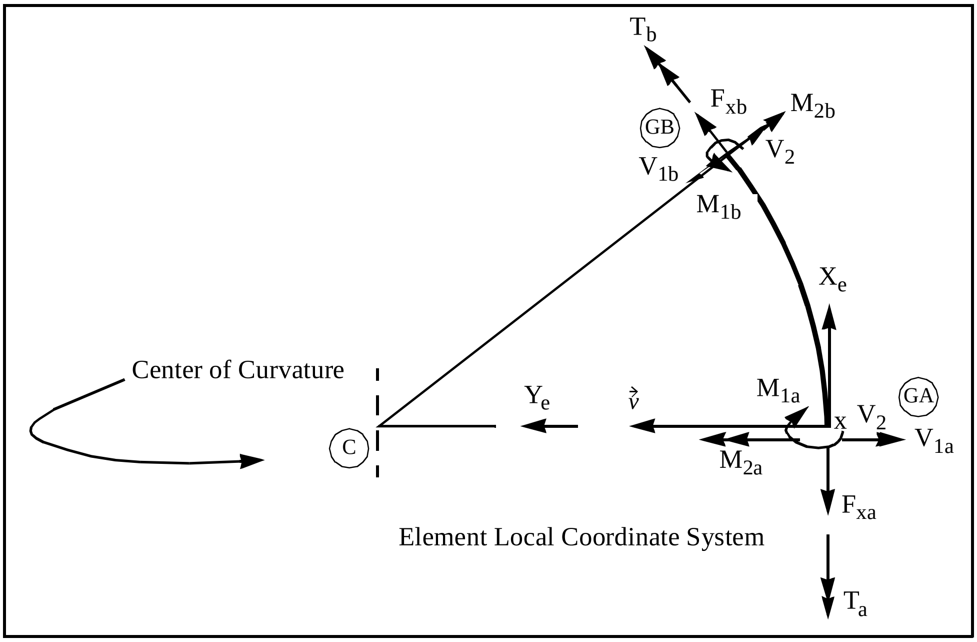


Figure 4-7 CELBOW Element Coordinate System

CHEXA - *Hexahedral Element Connection*

Description

Defines a six-sided solid element having from eight to twenty grid points.

Format

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example

CHEXA	110	7	3	8	12	13	14	9	
	5	4	16	19	20	17	23	27	
	31	32	33	28	25	24			

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSOLID property entry. (Integer >0)
Gi	Grid point identification numbers of connection points; each must be unique. (Integer > 0 or blank)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in counter-clockwise order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc. All corner grids must be defined, though midside grids are optional. See **Figure 4-8** for the GRID order.
3. The quadrilateral faces need not be planar.
4. Element stress output is given in the material coordinate system, which is provided on the PSOLID entry. The default for the material coordinate system is the BASIC coordinate system..
5. The second continuation is optional.

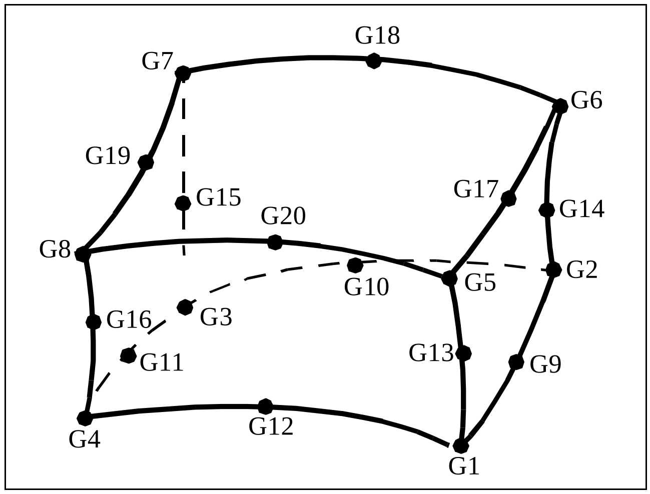


Figure 4-8 CHEXA grid point identification numbers

CMASS1 - *Scalar Mass Connection*

Description

Defines a scalar mass element.

Format

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2			

Example

CMASS1	32	6	2	1	2	3			

Field	Description
EID	**Element identification number. (Integer > 0)**
PID	**Identification number of a PMASS property entry. (Integer > 0 or blank; Default = EID)**
G1, G2	**Grid point identification number. (I**nteger ≥ 0 or blank)
C1, C2	**Degree of freedom component number. (0** ≤ **I**nteger ≤ 6 or blank)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded connection G1 or G2 with a corresponding blank or zero C1 or C2. If only scalar points and/or ground are involved, the CMASS3 entry is more efficient. (A grounded connection is a scalar point or grid point degree of freedom whose displacement is constrained to zero.)
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. The two connected degrees of freedom, (G1, C1) and (G2, C2), must be distinct.
5. To attach a scalar mass to a single degree of freedom, leave the second connected degree of freedom (G2, C2) blank

CMASS2 - *Scalar Mass Properties and Connection*

Description

Defines a scalar mass element without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	MASS	G1	C1	G2	C2			

Example

CMASS2	32	9.25	6	1	7				

Field	Description
EID	**Element identification number. (Integer > 0)**
MASS	Scalar mass **value. (Real)**
G1, G2	**Grid point identification number. (Integer ≥ 0 or blank)**
C1, C2	**Degree of freedom component number. (0 ≤ Integer ≤** 6 **or blank)**

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded connection G1 or G2 with a corresponding blank or zero C1 or C2. If only scalar points and/or ground are involved, the CMASS4 entry is more efficient. (A grounded connection is a scalar point or grid point degree of freedom whose displacement is constrained to zero.)
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. The two connected degrees of freedom, (G1, C1) and (G2, C2), must be distinct.
5. To attach a scalar mass to a single degree of freedom, leave the second connected degree of freedom (G2, C2) blank.

CMASS3 - *Scalar Mass Connection to Scalar Points*

Description

Defines a scalar mass element connected to scalar points only.

Format

1	2	3	4	5	6	7	8	9	10
CMASS3	EID	PID	S1	S2					

Example

CMASS3	13	42	62	1					

Field	Description
EID	**Element identification number. (Integer > 0)**
PID	**Identification number of a PMASS property entry. (Integer > 0 or blank; Default = EID)**
S1, S2	**Scalar point identification number. (Integer ≥ 0 or blank; S1 ≠** S2**)**

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. S1 or S2 may be blank or zero indicating a constrained degree of freedom.
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. To attach a mass to a single scalar degree of freedom, leave the second point, S2, blank.

CMASS4 - *Scalar Mass Properties and Connection to Scalar Points*

Description

Defines a scalar mass element connected to scalar points only and without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	MASS	S1	S2					

Example

CMASS4	23	14.92	6	23					

Field	Description
EID	**Element identification number. (Integer > 0)**
MASS	Scalar mass **value. (Real)**
S1, S2	**Scalar point identification number. (Integer ≥ 0 or blank; S1 ≠** S2**)**

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. S1 or S2 may be blank or zero indicating a constrained degree of freedom.
3. Scalar points defined on this entry will be automatically generated if not already specified on an SPOINT entry.
4. To attach a mass to a single scalar degree of freedom, leave the second point, S2, blank.

CONCT - *Substructure Connectivity*

Description

Defines the grid point and degree of freedom connectivities between two substructures for a manual COMBINE operation.

Format

1	2	3	4	5	6	7	8	9	10
CONCT	SID	C	SUBA	SUBB					
	GA1	GB1	GA2	GB2	GA3	GB3	GA4	GB4	
	GA5	GB5	-etc.-						

Example

CONCT	307	1236	WINGRT	FUSELAG					
	201	207	958	214	971	216	982		

Field	Description
SID	Connectivity set identification number. (Integer > 0)
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
SUBA, SUBB	Names of basic substructures being connected. (Character)
GAi, GBi	Grid or scalar point identification numbers. GAi from SUBA connects to GBi from SUBB via the degrees of freedom specified in C. (Integer > 0)

Remarks

1. At least one continuation entry must be present.
2. Components specified on a CONCT entry will be overridden by a RELES Bulk Data entry, if present.
3. SID may be shared across several CONCT and CONCT1 entries.
4. An alternate format is given by the CONCT1 entry.
5. Connectivity sets must be selected in the Substructure Control section (CONNECT = SID). Note that CONNECT is a subcommand of the substructure COMBINE command.
6. SUBA and SUBB must be component basic substructures of the pseudo structures being combined as specified on the substructure COMBINE command. SUBA and SUBB must not be components of the same pseudo structure.
7. If GTRAN has been invoked under the COMBINE command, the fields on CONCT and CONCT1 entries must be defined in terms of the revised coordinate system.

In **Figure 4-9**, a substructure tree and a set of substructure commands are shown. The CONNECT subcommand references the example shown in the preceding CONCT entry description. Here, pseudo structure PSUB1 and PSUB2 are combined and connected only at points in their respective basic component substructures WINGRT and FUSELAG.

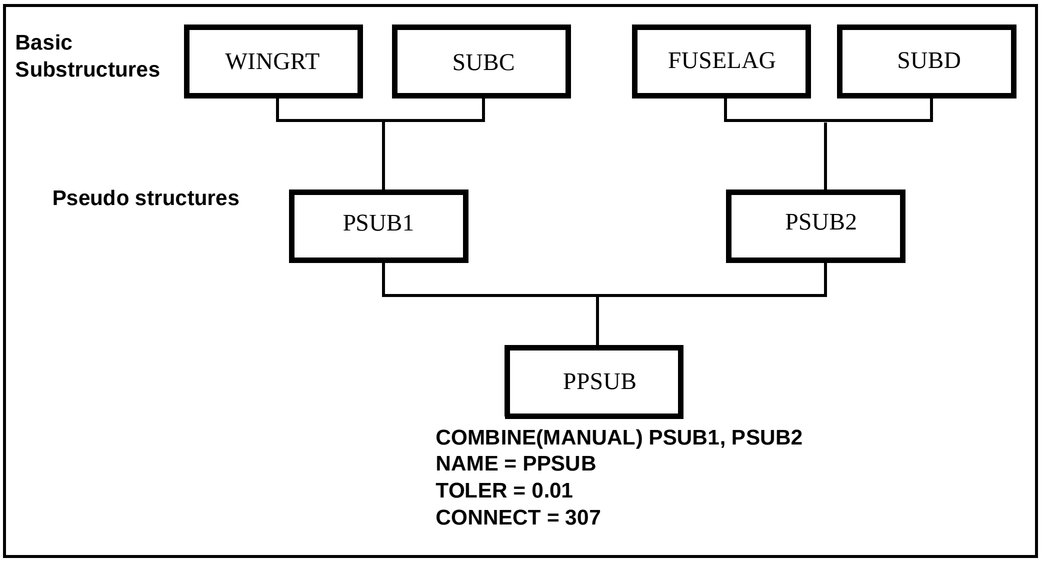


Figure 4-9

CONCT1 - *Substructure Connectivity*

Description

Defines the grid point and degree of freedom connectivities between two or more substructures for a manual COMBINE operation.

Format

1	2	3	4	5	6	7	8	9	10
CONCT1	SID	NAME1	NAME2	NAME3	NAME4	NAME5	NAME6	NAME7	
	C1	G11	G12	G13	G14	G15	G16	G17	
	C2	G21	G22	G23	G24	G25	G26	G27	
	C3	G31	G32	-etc.-					

Example

CONCT1	SID	WNGRT	FUSELAG	MDWNG	POD				
	123	528	17	32	106				
	46	518							

Field	Description
SID	Connectivity set identification number. (Integer > 0)
NAMEj	Basic substructure name. (Character)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Gij	Grid or scalar point identification numbers in substructure, NAMEj, with components Ci. (Integer > 0)

Remarks

1. At least one continuation entry must be present.
2. Components specified on CONCT1 entries will not be overridden by data supplied on RELES entries.
3. SID may be shared across several CONCT and CONCT1 entries.
4. An alternate format is given by the CONCT entry.
5. Connectivity sets must be selected in the Substructure Control section (CONNECT = SID). Note that CONNECT is a subcommand of the substructure COMBINE command.
6. The NAMEj's must be the names of basic substructure components of the pseudo structures named on the COMBINE entry in the Substructure Control section. See the CONCT entry for a more complete discussion related to the combination of two substructures.
7. CONCT1 and its continuations effectively describe a map of connectivities. Grid points entered in the corresponding field of a substructure name define the connectivity participation for that substructure. Each continuation defines the connection relationships among the participating substructures for the components entered.
8. If GTRAN has been invoked under the COMBINE command, the fields on CONCT and CONCT1 entries must be defined in terms of the revised coordinate system.

CONM1 - *Concentrated Mass Element Connection*

Description

Defines a 6x6 symmetric mass matrix at a grid point.

Format

1	2	3	4	5	6	7	8	9	10
CONM1	EID	G	CID	M11	M21	M22	M31	M32	
	M33	M41	M42	M43	M44	M51	M52	M53	
	M54	M55	M61	M62	M63	M64	M65	M66	

Example

CONM1	2	22	2	2.9		6.3			
	4.8				28.6				
		28.6						28.6	

Field	Description
EID	Element identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number for mass matrix definition. (Integer ≥ 0 or blank; Default = 0)
Mij	Mass matrix values. (Real; Default = 0.0)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. See the **CONM2** entry for an alternate means of specifying a diagonal mass matrix.

CONM2 - *Concentrated Mass Element Connection*

Description

Defines a concentrated mass at a grid point.

Format

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

Example

CONM2	2	15	6	49.7					
	16.2		16.2			7.8			

Field	Description
EID	Element identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number for mass matrix definition. (Integer ≥ 0, blank, or -1; Default = 0)
M	Mass value. (Real)
X1,X2,X3	Offset distances for the mass in the CID coordinate system. See Remark 4. (Real or blank; Default = 0.0)
Iij	Mass moments of inertia measured at the mass c.g. in coordinate system CID. (Real or blank; Default = 0.0).

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. For a more general means of defining concentrated mass at grid points, see the **CONM1** entry.
3. The continuation entry is not required.
4. If CID = -1, then X1, X2, X3 are the coordinates of the mass center of gravity expressed in the basic coordinate system.
5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & 0 & 0 & 0 & 0 & 0 \\ 0 & M & 0 & 0 & 0 & 0 \\ 0 & 0 & M & & & \\ 0 & 0 & 0 & I_{11} & -I_{21} & -I_{31} \\ 0 & 0 & 0 & -I_{21} & I_{22} & -I_{32} \\ 0 & 0 & 0 & -I_{31} & -I_{32} & I_{33} \end{bmatrix}$$

where:

$$\begin{aligned} I_{11} &= M(X_2^2 + X_3^2) \\ I_{22} &= M(X_3^2 + X_1^2) \\ I_{33} &= M(X_1^2 + X_2^2) \\ I_{21} &= MX_2X_1 \\ I_{31} &= MX_3X_1 \\ I_{32} &= MX_3X_2 \end{aligned}$$

The form of the inertia matrix at the grid point is:

$$\begin{bmatrix} M & 0 & 0 & 0 & MX_3 & -MX_2 \\ 0 & M & 0 & -MX_3 & 0 & 0 \\ 0 & 0 & M & MX_2 & -MX_1 & 0 \\ 0 & -MX_3 & MX_2 & \bar{I}_{11} & -\bar{I}_{21} & -\bar{I}_{31} \\ MX_3 & 0 & -MX_1 & -\bar{I}_{21} & \bar{I}_{22} & -\bar{I}_{32} \\ -MX_2 & 0 & 0 & -\bar{I}_{31} & -\bar{I}_{32} & \bar{I}_{33} \end{bmatrix}$$

where:

$$\begin{aligned} \bar{I}_{11} &= I_{11} + M(X_2^2 + X_3^2) \\ \bar{I}_{22} &= I_{22} + M(X_3^2 + X_1^2) \\ \bar{I}_{33} &= I_{33} + M(X_1^2 + X_2^2) \\ \bar{I}_{21} &= I_{21} + MX_2X_1 \\ \bar{I}_{31} &= I_{31} + MX_3X_1 \\ \bar{I}_{32} &= I_{32} + MX_3X_2 \end{aligned}$$

Mass matrices are positive definite. A warning message is issued if the Inertia matrix at the reference grid point is not positive definite.

CONROD - *Rod Element Properties and Connection*

Description

Defines a rod element without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

Example

CONROD	2	16	17	23	2.69				

Field	Description
EID	Element identification number. (Integer > 0)
G1, G2	Grid point identification number. (Integer > 0, G1 ≠ G2)
MID	Material property identification number. See Remarks 2 and 3. (Integer > 0)
A	Cross sectional area. (Real; Default = 0.0)
J	Torsional constant. (Real; Default = 0.0).
C	Stress recovery distance for torsional shear stress calculation. (Real; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real; Default = 0.0)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. In structural problems, the CONROD entry may only reference a MAT1 material property entry..

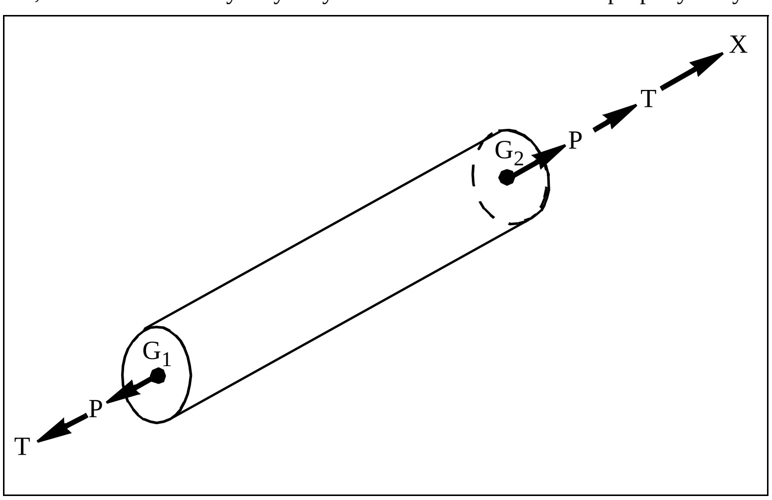


Figure 4-10 CONROD Element Forces and Moments

CORD1C - *Cylindrical Coordinate System Definition, Grid Point Form*

Description

Defines a cylindrical coordinate system by reference to three grid points.

Format

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example

CORD1C	3	16	32	19					

Field	Description
CIDA, CIDB	Coordinate system identification numbers. (Integer > 0)
GiA, GiB	Grid point identification numbers. See Remark 3. (Integer > 0)

Remarks

1. Up to two coordinate systems may be defined on this entry.
2. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S entries must be unique.
3. The three grid points G1, G2, and G3 must be unique, non-colinear, and must be defined in coordinate systems whose definition does not involve the coordinate system being defined. The first point is the origin, the second lies on the positive z-axis, and the third lies in the plane of the azimuthal origin.
4. The location of a grid point in this coordinate system is given by (R, θ , Z) where θ is measured in degrees.
5. The displacement components at P, (u_R, u_θ, u_z) , are dependent on the location of P as shown in **Figure 4-11**.
6. Points on the z-axis may not have their displacement directions defined in this coordinate system since an ambiguity results.

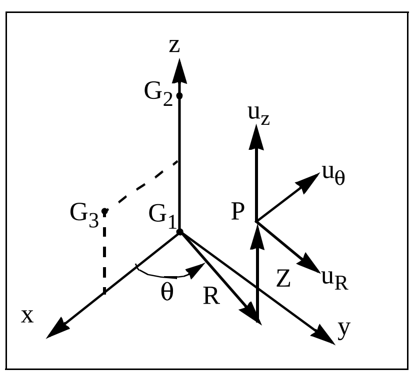


Figure 4-11 CORD1C coordinate system

CORD1R - *Rectangular Coordinate System Definition, Grid Point Form*

Description

Defines a rectangular coordinate system by reference to three grid points.

Format

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example

CORD1R	101	11	12	13					

Field	Description
CIDA, CIDB	Coordinate system identification numbers. (Integer > 0)
GiA, GiB	Grid point identification numbers. See Remark 3. (Integer > 0)

Remarks

1. Up to two coordinate systems may be defined on this entry.
2. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S entries must be unique.
3. The three grid points G1, G2, and G3 must be unique, non-colinear, and must be defined in coordinate systems whose definition does not involve the coordinate system being defined. The first point is the origin, the second lies on the positive z-axis, and the third lies in the x-z plane.
4. The location of a grid point in this coordinate system is given by (X, Y, Z).
5. The displacement components at P are given by (u_x, u_y, u_z), as shown in **Figure 4-12**.

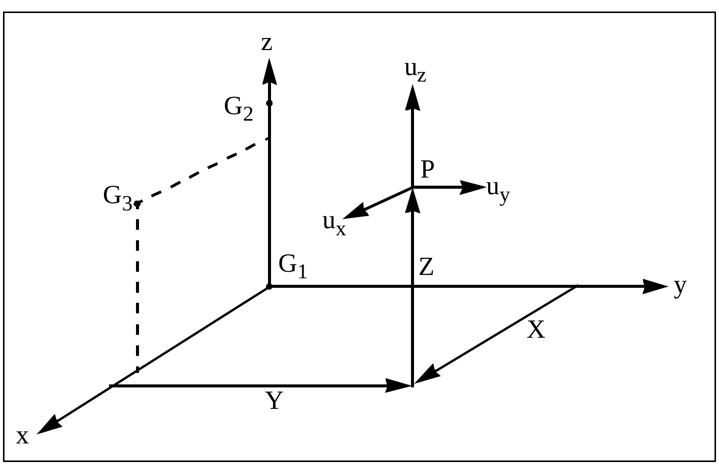


Figure 4-12 CORD1R coordinate system

CORD1S - *Spherical Coordinate System Definition, Grid Point Form*

Description

Defines a spherical coordinate system by reference to three grid points.

Format

1	2	3	4	5	6	7	8	9	10
CORD1S	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example

CORD1S	11	1	2	3					

Field	Description
CIDA, CIDB	Coordinate system identification numbers. (Integer > 0)
GiA, GiB	Grid point identification numbers. See Remark 3. (Integer > 0)

Remarks

1. Up to two coordinate systems may be defined on this entry.
2. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S entries must be unique.
3. The three points G1, G2, and G3 must be unique, non-colinear, and must be defined in coordinate systems whose definition does not involve the coordinate system being defined. The first point is the origin, the second lies on the positive z-axis, and the third lies in the plane of the azimuthal origin.
4. The location of a grid point in this coordinate system is given by (R,Θ,Φ,) where Θ and Φ are measured in degrees.
5. The displacement components at P, (u_r, u_θ, u_ϕ) , are dependent on the location of P as shown in **Figure 4-13**.
6. Points on the polar axis may not have their displacement directions defined in this coordinate system since an ambiguity results.

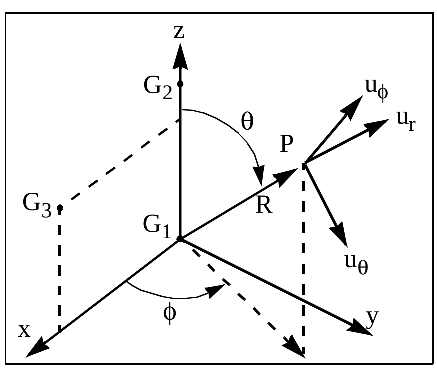


Figure 4-13 CORD1S coordinate system

CORD2C - *Cylindrical Coordinate System Definition, Component Form*

Description

Defines a cylindrical coordinate system by reference to the coordinates of three points.

Format

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example

CORD2C	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Description
CID	Coordinate system identification number. (Integer > 0)
RID	Identifier of a reference coordinate system not dependent on CID. (Integer ≥ 0 or blank; Default = 0)
Ai, Bi, Ci	Coordinates of three points in the RID coordinate system. (Real)

Remarks

1. The continuation entry is required.
2. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S entries must be unique.
3. The three points (A1, A2, A3), (B1, B2, B3), and (C1, C2, C3) must be unique and non-colinear. The first point is the origin, the second lies on the positive z-axis, and the third lies in the plane of the azimuthal origin.
4. An RID of zero or blank references the basic coordinate system.
5. The location of a grid point in this coordinate system is given by (R, θ , Z) where θ is measured in degrees.
6. The displacement components at P, (u_r, u_θ, u_z) , are dependent on the location of P as shown in **Figure 4-14**.
7. Points on the z-axis may not have their displacement direction defined in this coordinate system, since an ambiguity results.

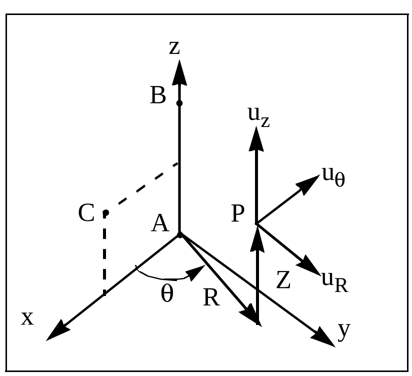


Figure 4-14 CORD2C coordinate system

CORD2R - *Rectangular Coordinate System Definition, Component Form*

Description

Defines a rectangular coordinate system by reference to the coordinates of three points.

Format

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example

CORD2R	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Description
CID	Coordinate system identification number. (Integer > 0)
RID	Identifier of a reference coordinate system not dependent on CID. (Integer ≥ 0 or blank; Default = 0)
Ai, Bi, Ci	Coordinates of three points in the RID coordinate system. (Real)

Remarks

1. The continuation entry is required.
2. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S entries must be unique.
3. The three points (A1, A2, A3), (B1, B2, B3), and (C1, C2, C3) must be unique and non-colinear. The first point is the origin, the second lies on the positive z-axis, and the third lies in the x-z plane.
4. An RID of zero or blank references the basic coordinate system.
5. The location of a grid point in this coordinate system is given by (X, Y, Z).
6. The displacement components at P are given by (u_x, u_y, u_z) , as shown in **Figure 4-15**.

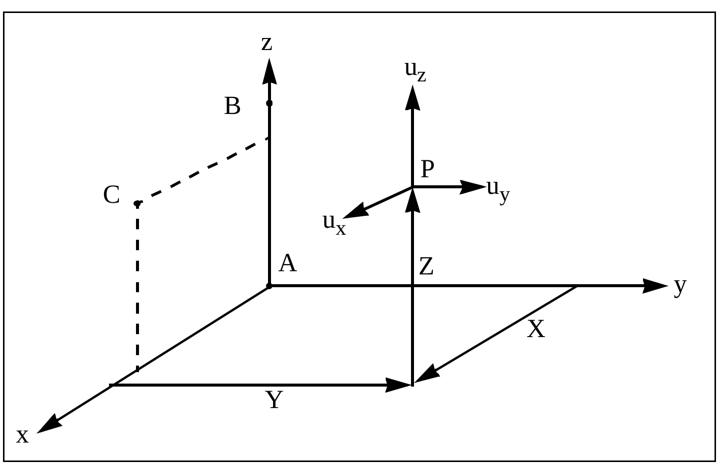


Figure 4-15 CORD2R coordinate system

CORD2S - *Spherical Coordinate System Definition, Component Form*

Description

Defines a spherical coordinate system by reference to the coordinates of three points.

Format

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example

CORD2S	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Description
CID	Coordinate system identification number. (Integer > 0)
RID	Identifier of a reference coordinate system not dependent on CID. (Integer ≥ 0 or blank; Default = 0)
Ai, Bi, Ci	Coordinates of three points in the RID coordinate system. (Real)

Remarks

1. The continuation entry is required.
2. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S entries must be unique.
3. The three points (A1, A2, A3), (B1, B2, B3), and (C1, C2, C3) must be unique and non-colinear. The first point is the origin, the second lies on the positive z-axis, and the third lies in the plane of the azimuthal origin.
4. An RID of zero or blank references the basic coordinate system.
5. The location of a grid point in this coordinate system is given by (R,Θ,Φ) where Θ and Φ are measured in degrees.
6. The displacement components at P, (u_r, u_θ, u_ϕ) , are dependent on the location of P as shown in **Figure 4-16**.
7. Points on the z-axis may not have their displacement direction defined in this coordinate system, since an ambiguity results.

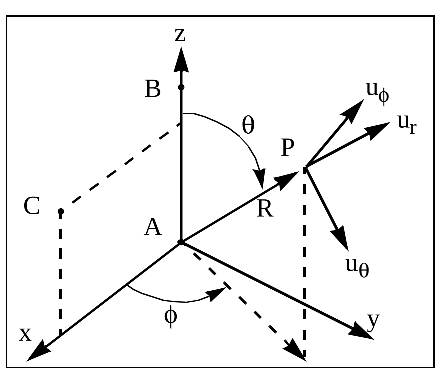


Figure 4-16 CORD2S coordinate system

CPENTA - *Five-Sided Solid Element Connection*

Description

Defines a solid element with six to fifteen grid points.

Format

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example

CPENTA	110	7	3	8	12	13	14	9	
	5	4	16	19	21	24	37	41	
	5								

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSOLID property entry. (Integer > 0)
Gi	Grid point identification numbers of connection points; G1 through G4 must be unique. (Integer > 0)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. There is no nonstructural mass.
3. Element stress output is given in the material coordinate system, which is provided on the PSOLID entry. The default for the material coordinate system is the BASIC coordinate system

...

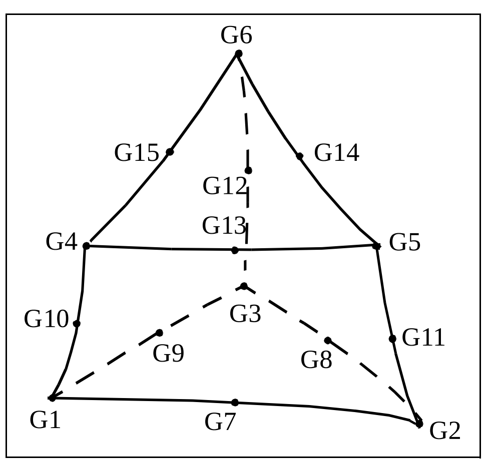


Figure 4-17 CPENTA Element Connectivity

4. The CPENTA element coordinate system orientation is determined as follows: The origin is located at the midpoint of the straight line segment connecting grid points G1 and G4. The z axis points in the direction of the G4-G5-G6 face, and is oriented somewhere between a line joining the triangular face centroids and a line perpendicular to the midplane. (The midplane is defined as the surface passing through the midpoints of the lines connecting opposing triangular face grid points.) The x and y axes are chosen such that, together with z, they form a regular cartesian system, with positive x pointing in the direction of the G2-G5 edge, and positive y pointing in the direction of the G3-G6 edge.,

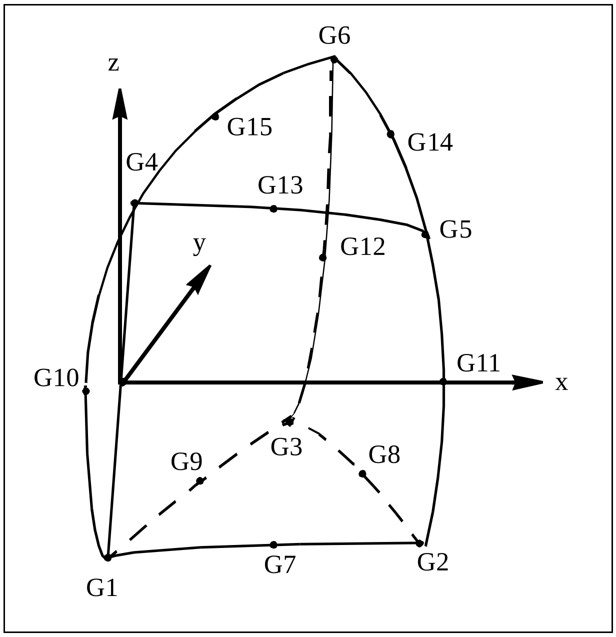


Figure 4-18 CPENTA Element Coordinate System

CQUAD4 - *Quadrilateral Shell Element Connection*

Description

Defines an isoparametric membrane-bending quadrilateral shell element with variable element thickness and layered composite material analysis capabilities.

Format

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
			T1	T2	T3	T4			

Example

CQUAD4	101	17	1001	1005	1010	1024	45.0	0.01	
			0.03	0.125	0.05	0.04			

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSHELL property entry or a PCOMP, PCOMP1, or PCOMP2 entry for composites. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integer > 0; All Gi unique)
THETA	Material property orientation specification, given as a real number in degrees. (Real or blank; Default = 0.0)
MCID	Coordinate system ID for material property orientation specification. The element material x-axis is determined by the projection of the MCID x-axis onto the plane of the element. (Integer ≥ 0 or blank (basic coordinate system is indicated by MCID = 0); Default THETA = 0.0)
ZOFFS	Element reference plane (element mid-plane) offset from the plane of grid points. See Remark 3. (Real ≥ 0.0 or blank; Default = 0.0)
Ti	Membrane thickness of element at grid points Gi. See Remark 4. (Real or blank)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.

2. As shown in **Figure 4-19**, grid point ordering (which must be consecutive around the element) determines the orientation of the local element coordinate system, and the element edge from which the material orientation is measured when the THETA option is used.

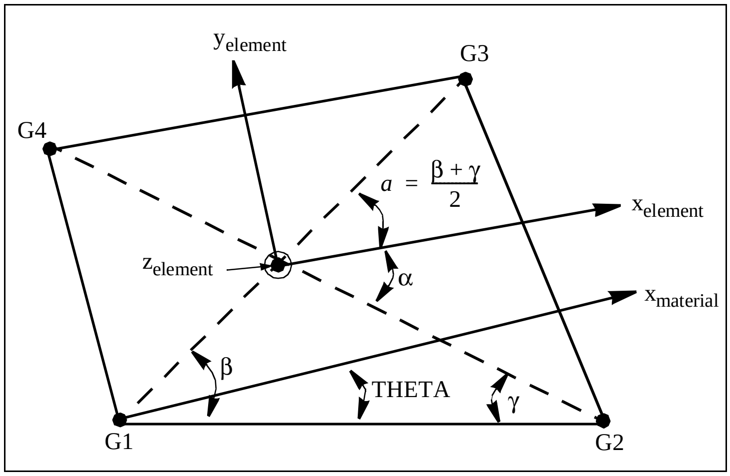


Figure 4-19 CQUAD4 Element Geometry

3. When the MCID material orientation option is used, the orientation of the local material coordinate system is determined by the projection of the MCID x-axis onto the plane of the element as shown in **Figure 4-20**. The material coordinate system (THETA or MCID) and the offset (ZOFFS) may also be specified on the PSHELL entry, and provide defaults if the corresponding CQUAD4 entry fields are blank.

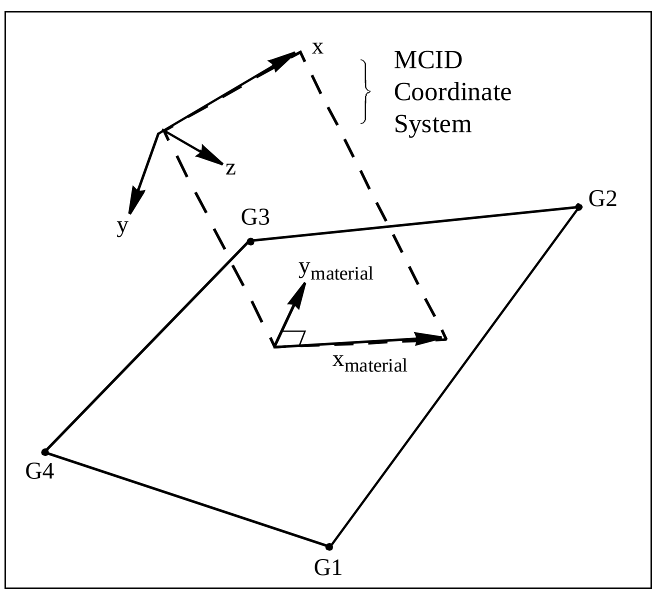


Figure 4-20 CQUAD4 Material Coordinate System, MCID Option

4. The Ti fields are optional and default to the value of T specified on the PSHELL entry. If all Ti are blank, the continuation entry is not required.

CROD - *Rod Element Connection*

Description

Defines a tension-compression-torsion rod element.

Format

1	2	3	4	5	6	7	8	9	10
CROD	EIDA	PIDA	G1A	G2A	EIDB	PIDB	G1B	G2B	

Example

CROD	12	13	21	23	3	12	24	5	

Field	Description
EIDi	Element identification number. (Integer > 0)
PIDi	Identification number of a PROD property entry. (Integer > 0; Default = EID)
G1i, G2i	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. See CONROD for an alternative method of rod element definition, without reference to a property entry.
3. One or two CROD elements may be defined on a single entry.

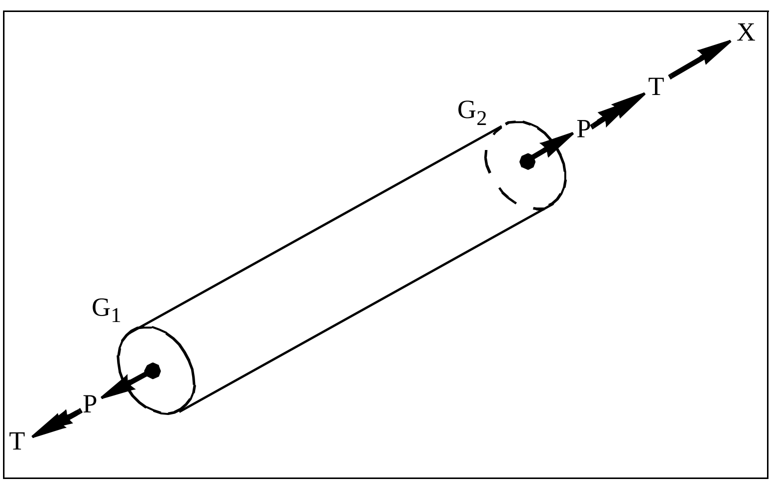


Figure 4-21 CROD Element Forces

CSHEAR - *Shear Panel Element Connection*

Description

Defines a shear panel element.

Format

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4			

Example

CSHEAR	3	6	1	5	3	7			

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSHEAR property entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 through G4 unique)

Remarks

1. Each element identification number must be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.

3. All interior angles must be less than 180 degrees.

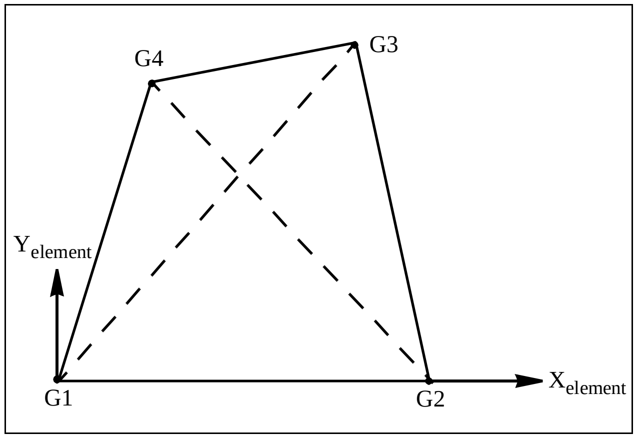


Figure 4-22 CSHEAR Element Coordinate System

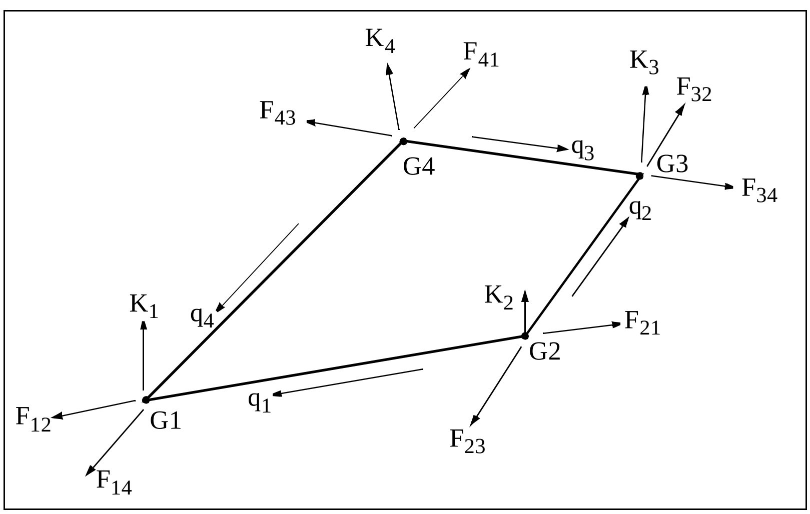


Figure 4-23 CSHEAR Element Forces

CTETRA - *Tetrahedral Solid Element Connection*

Description

Defines a four-sided solid element having from four to ten grid points.

Format

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

Example

CTETRA	110	7	3	8	12	13	14	9	
	5	4	16	19					

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSOLID property entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer >0; G1 through G4 must be unique)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Any or all of the midside nodes may be deleted. The continuation entry is not required when G7 through G10 do not exist.
3. There is no nonstructural mass.

4. Output stresses are given in basic coordinate system.

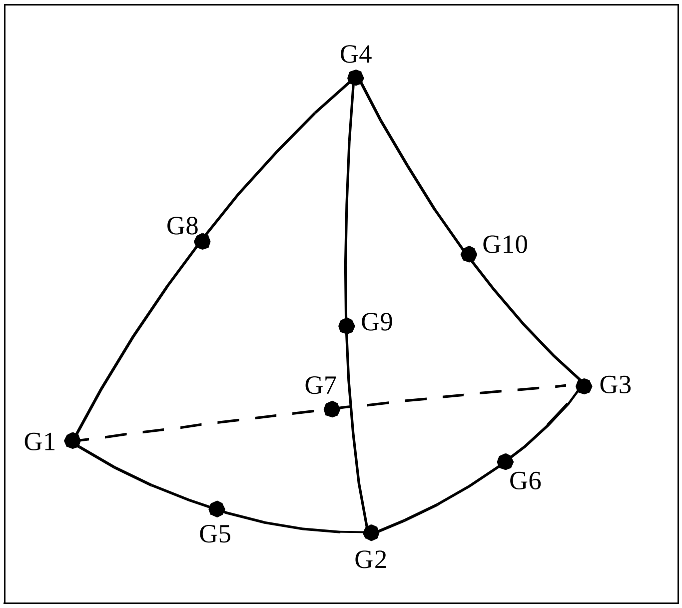


Figure 4-24 CTETRA Connectivity

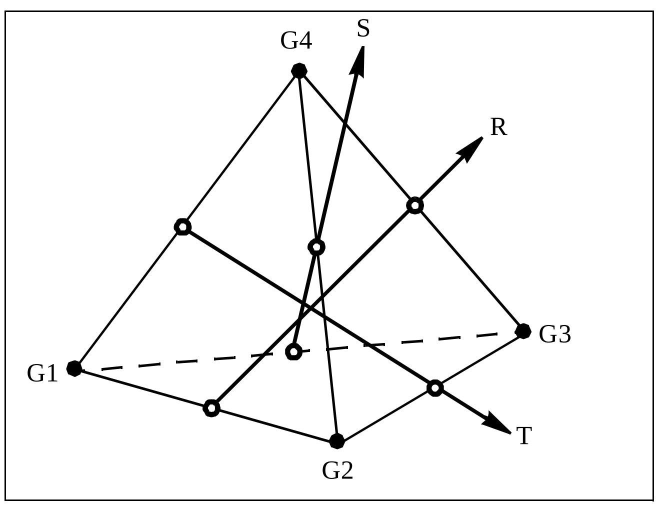


Figure 4-25 CTETRA Element Coordinate System

CTORDRG - *Toroidal Ring Element Connection*

Description

Defines an axisymmetric toroidal cross-section ring element.

Format

1	2	3	4	5	6	7	8	9	10
CTORDRG	EID	PID	G1	G2	A1	A2			

Example

CTORDRG	25	2	47	48	30.0	60.0			

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PTORDRG property entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)
A1	Angle of curvature at grid point 1, measured in degrees. See **Figure 4-26**. (Real; 0. ≤ A1 ≤ 180. ; A2 ≥ A1)
A2	Angle of curvature at grid point 2, measured in degrees. See **Figure 4-26**. (Real; 0. ≤ A2 ≤ 180. ; A2 ≥ A1)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1 and G2 must lie in the x-z plane of the basic coordinate system and to the right of the axis of symmetry (the z-axis).
3. If A1 = 0., the element is assumed to be a shell cap.
4. Only elements of zero or positive Gaussian curvature may be used.

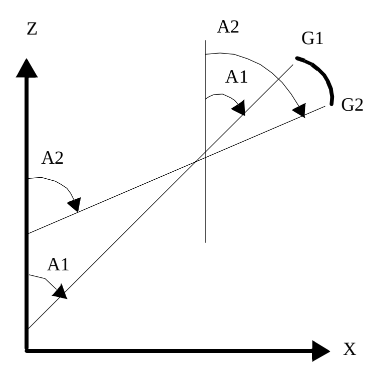


Figure 4-26 CTORDRG Element Geometry

CTRAPAX - *Trapezoidal Ring Element Connection*

Description

Defines an axisymmetric trapezoidal cross-section ring element with non-axisymmetric deformation.

Format

1	2	3	4	5	6	7	8	9	10
CTRAPAX	EID	PID	R1	R2	R3	R4	TH		

Example

CTRAPAX	15	5	10	11	12	13	30.0		

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PTRAPAX property entry. (Integer > 0)
Ri	Axisymmetric ring (RINGAX) identification numbers of connection points. (Integer >0; R1 through R4 unique)
TH	Material property orientation angle in degrees. See **Figure 4-27** for the TH sign convention. (Real)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. CTRAPAX entries are allowed only if an AXIC entry is also present.
3. RINGAX identification numbers R1, R2, R3, and R4 must be ordered counterclockwise around the perimeter of the element.
4. For a discussion of the axisymmetric ring problem, see Section 5.11 of the Theoretical Manual.
5. The lines connecting R1 to R2 and R4 to R3 must be parallel to the r axis.
6. This element cannot be modeled with a grid point on the axis of symmetry.

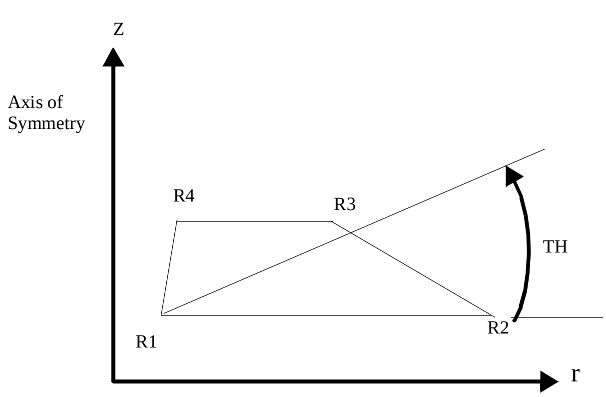


Figure 4-27 CTRAPAX sign convention for TH

CTRAPRG - *Trapezoidal Ring Element Properties and Connection*

Description

Defines an axisymmetric trapezoidal cross-section ring element without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CTRAPRG	EID	G1	G2	G3	G4	TH	MID		

Example

CTRAPRG	72	13	14	15	16	29.2	13		

Field	Description
EID	Element identification number. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 through G4 unique)
TH	Material property orientation angle in degrees. See **Figure 4-28** for the TH sign convention. (Real)
MID	Material property identification number. (Integer > 0)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The four grid points must lie in the x-z plane of both the basic and any local coordinate systems and to the right of the axis of symmetry (the z-axis), except that the grid points G1 and G4 may lie on the axis of symmetry in the limiting case when the element becomes a solid core element. (See **Section 1.3.7.1**.)
3. Grid points G1 through G4 must be ordered counterclockwise around the perimeter of the element.
4. The line connecting grid points G1 and G2 and the line connecting grid points G3 and G4 must be parallel to the x-axis.
5. All interior angles must be less than 180 degrees.
6. For structural problems, the material property identification number must reference either a MAT1 or MAT3 entry..

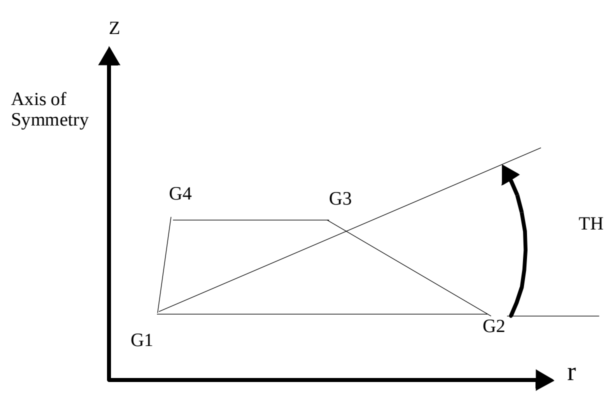


Figure 4-28 CTRAPRG grid point identification numbers.

CTRIA3 - *Triangular Shell Element Connection*

Description

Defines an isoparametric membrane-bending triangular shell element with variable element thickness and layered composite material analysis capabilities.

Format

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
			T1	T2	T3				

Example

CTRIA3	101	17	1001	1005	1010	45.0	0.01		
			0.03	0.125	0.05				

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PSHELL property entry or a PCOMP, PCOMP1, or PCOMP2 entry for composites. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integer > 0; All Gi unique)
THETA	Material property orientation specification, given as a real number in degrees. (Real or blank; Default = 0.0)
MCID	Coordinate system ID for material property orientation specification. The element material x-axis is determined by the projection of the MCID x-axis onto the plane of the element. (Integer ≥ 0 or blank; Default THETA = 0.0)
ZOFFS	Element reference plane (element mid-plane) offset from the plane of grid points. See Remark 3. (Real ≥ 0.0 or blank; Default = 0.0)
Ti	Membrane thickness of element at grid points Gi; see Remark 4 for default.

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. As shown in **Figure 4-29**, grid point ordering determines the orientation of the local element coordinate system, and the element edge from which the material orientation is measured when the THETA option is used.
3. When the MCID material orientation option is used, the orientation of the local material coordinate system is determined by the projection of the MCID x-axis onto the plane of the element as shown in **Figure 4-30**. The material coordinate system (THETA or MCID) and the offset (ZOFFS) may also be specified on the PSHELL entry, and provide defaults if the corresponding CTRIA3 entry fields are blank.
4. The Ti fields are optional and default to the value of T specified on the PSHELL entry. If all Ti are blank, the continuation entry is not required.

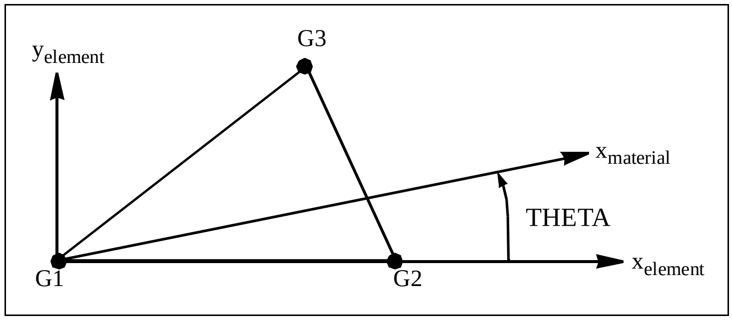


Figure 4-29 CTRIA3 Element Geometry

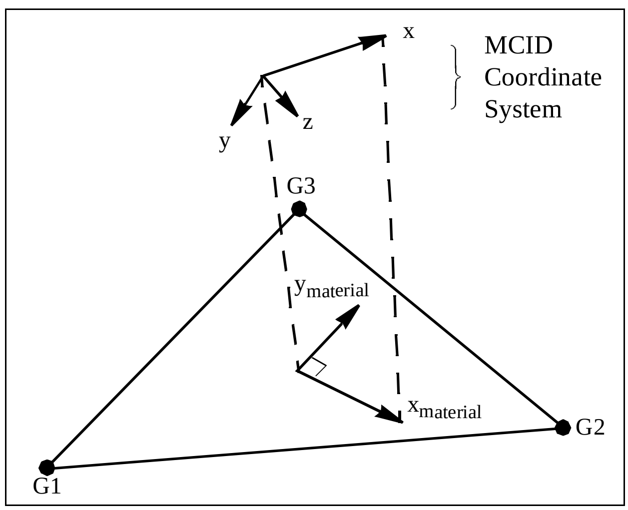


Figure 4-30 CTRIA3 Material Coordinate System, MCID Option

CTRIAAX - *Triangular Ring Element Connection*

Description

Defines an axisymmetric triangular cross-section ring element with non-axisymmetric deformation.

Format

1	2	3	4	5	6	7	8	9	10
CTRIAAX	EID	PID	R1	R2	R3	TH			

Example

CTRIAAX	20	15	42	43	52	60.0			

Field	Description
EID	Element identification number. (Integer > 0)
PID	Identification number of a PTRIAAX property entry. (Integer > 0)
Ri	Axisymmetric ring (RINGAX) identification numbers of connection points. (Integer >0; R1 through R3 unique)
TH	Material property orientation angle in degrees. See **Figure 4-31** for the TH sign convention. (Real)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The CTRIAAX entry is allowed only if an AXIC entry is also present.
3. .RINGAX identification numbers R1, R2, and R3 must be ordered counterclockwise around the perimeter.
4. For a discussion of the axisymmetric ring problem, see Section 5.11 of the Theoretical Manual.

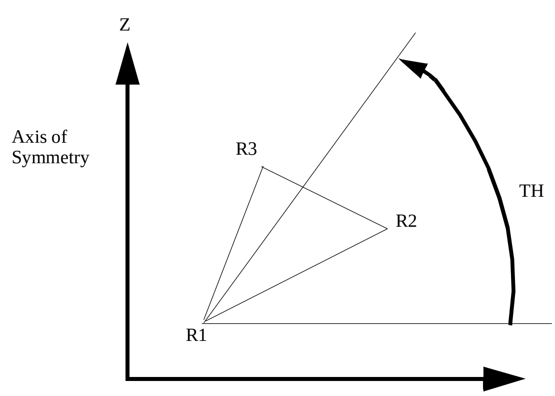


Figure 4-31 CTRIAAX sign convention for TH

CTRIARG - *Triangular Ring Element Properties and Connection*

Description

Defines an axisymmetric triangular cross section ring element without reference to a property entry.

Format

1	2	3	4	5	6	7	8	9	10
CTRIARG	EID	G1	G2	G3	TH	MID			

Example

CTRIARG	16	12	13	14	29.2	17			

Field	Description
EID	Element identification number. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 through G3 unique)
TH	Material property orientation angle in degrees. See **Figure 4-32** for the TH sign convention. (Real)
MID	Material property identification number. (Integer > 0)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The grid points must lie in the x-z plane of both the basic and any local coordinate systems and to the right of the axis of symmetry (the z-axis).
3. Grid points G1, G2, and G3 must be ordered counterclockwise around the perimeter of the element as shown.
4. For structural problems, the material property identification number must reference either a MAT1 or MAT3 entry..

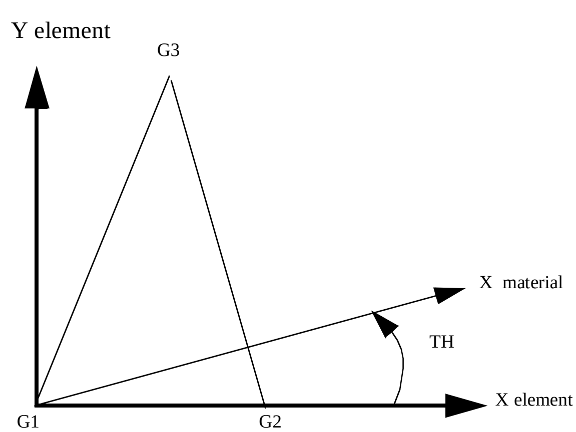


Figure 4-32 CTRIARG grid point identification numbers

CTUBE - *Tube Element Connection*

Description

Defines a tension-compression-torsion thin-walled tube element.

Format

1	2	3	4	5	6	7	8	9	10
CTUBE	EIDA	PIDA	G1A	G2A	EIDB	PIDB	G1B	G2B	

Example

CTUBE	12	13	21	23	3	12	24	5	

Field	Description
EIDi	Element identification number. (Integer > 0)
PIDi	Identification number of a PTUBE property entry. (Integer > 0; Default = EID)
G1i, G2i	Grid point identification numbers of connection points. (Integer > 0; G1i ≠ G2i

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Up to two CTUBE elements may be defined on a single entry.

CVISC - *Viscous Damper Element Connection*

Description

Defines a viscous damper element.

Format

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2	EID	PID	G1	G2	

Example

CVISC	21	6327	29	31	22	6527	35	33	

Field	Description
EID	Element identification number.
PID	Identification number of PVISC property entry. (Integer > 0; Default = EID).
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Up to two CVISC elements may be defined on a single entry.
3. Viscous damper elements are applicable in direct dynamics (i.e., non-modal) problems only.

CYJOIN - *Cyclic Symmetry Boundary Points*

Description

Defines the boundary points of a segment for cyclic symmetry analysis.

Format

1	2	3	4	5	6	7	8	9	10
CYJOIN	SIDE	C	G1	G2	G3	G4	G5	G6	
	G7	-etc.-							

Example

CYJOIN	1		7	9	16	25	33	64	
	72								

Alternate Format and Example

CYJOIN	SIDE	C	G1	“THRU”	G2				
CYJOIN	2	S	6	THRU	32				

Field	Description
SIDE	Side identification number. (Integer 1 or 2)
C	Coordinate system type. See Remark 3. (Character “R”, “C”, “S”, or blank)
Gi	Grid or scalar point identification numbers. (Integer > 0)

Remarks

1. CYJOIN entries are only used in cyclic symmetry problems. The type of symmetry (rotational or dihedral) is determined by the value of CTYPE, specified on a Bulk Data PARAM entry.
2. For rotational symmetry problems there must be one logical entry for side 1 and one for side 2. Since the two CYJOIN grid lists specify grid points to be connected, both lists must have the same length.
3. For dihedral symmetry problems, side 1 refers to the boundary between segments and side 2 refers to the middle of a segment. A coordinate system must be referenced in field 3, where R = rectangular, C = cylindrical, and S = spherical.
4. All components of displacement at boundary points are connected to adjacent segments, except those constrained by SPC, MPC, or OMIT entries.

DAREA - *Dynamic Load Scale Factor*

Description

The DAREA entry is used in conjunction with the RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries to define the load factor, A, at a specific degree of freedom at which the dynamic load is to be applied.

Format

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

Example

1	2	3	4	5	6	7	8	9	10
DAREA	3	6	2	8.2	15	1	10.1		

Field	Description
SID	Set identification number. (Integer > 0)
Pi	GRID, SPOINT or EPOINT identification number of a degree of freedom in the p-set. (Integer > 0)
Ci	Component number associated with Pi. (integer 1 throgh 6 for Grid point; balnk or zero for scalar and extra point.)
Ai	Scale (area) factor A for the designated component. (Real)

Remarks

1. Up to two scale factors may be defined on a single entry.
2. DAREA must be referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1, or TLOAD2 entry which in turn is selected by a DLOAD Case Control directive. Static loads can also be used to generate load coefficients for g-set degrees of freedom in dynamics. (See decription of RLOAD and TLOAD-type entries.)
3. For axisymmetric problems, Gi represents the NASTRAN (internal) grid ID and is given by the following algorithm: G = user-supplied (external) ring ID + 10^6 x (harmonic + 1).

DAREAS - *Dynamic Load Scale Factor, Substructure Analysis*

Description

The DAREAS entry is used in conjunction with the RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries to define the point where the dynamic load is to be applied with the scale (area) factor A.

Format

1	2	3	4	5	6	7	8	9	10
DAREAS	SID	NAME	G1	C1	A1	G2	C2	A2	

Example

DAREAS	3	SKIN	6	2	8.2	15	1	10.1	

Field	Description
SID	DAREA set identification number. (Integer > 0)
NAME	Basic substructure name. (Character)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
Ai	Scale (area) factor A for the designated component. (Real)

Remarks

1. Up to two scale factors may be defined on a single entry.
2. DAREAS scale factors are used in substructure SOLVE operations.
3. Points referenced on DAREAS entries must exist in the SOLVEd structure.

DEFORM - *Element Static Deformation*

Description

Defines static enforced axial deformation for one-dimensional elements.

Format

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID1	D1	EID2	D2	EID3	D3		

Example

DEFORM	1	535	0.05	536	-0.10				

Field	Description
SID	Deformation set identification number. (Integer > 0)
EIDi	Element identification number. (Integer > 0)
Di	Deformation. (Real; positive value indicates elongation)

Remarks

1. The referenced element must be one-dimensional (CBAR, CONROD, CROD or CTUBE.)
2. Element deformation sets are selected by the DEFORM Case Control directive in static analysis.
3. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
4. Up to three element enforced deformations may be defined on a single entry.

DELAY - *Dynamic Load Time Delay*

Description

Defines the time delay term, τ, in the equations of frequency or transient dynamic loading.

Format

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P1	C1	T1	P2	C2	T2		

Example

DELAY	5	21	6	4.25	7	6	8.1		

Field	Description
SID	DELAY set identification number. See Remark 2. (Integer > 0)
Pi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
Ti	Time delay, τ, for the designated grid/coordinate pair. (Real)

Remarks

1. Up to two dynamic load time delays may be defined on a single entry.
2. The DELAY set identification number, SID, must be referenced on an RLOAD1 or RLOAD2 entry for frequency response, or a TLOAD1 or TLOAD2 entry for transient response in order to be used by NASTRAN-CORE.

DELAYS - *Dynamic Load Time Delay, Substructure Analysis*

Description

Defines the time delay term, τ, in equations for frequency or transient dynamic loading applied to a substructure.

Format

1	2	3	4	5	6	7	8	9	10
DELAYS	SID	NAME	P1	C1	T1	P2	C2	T2	

Example

DELAYS	5	SKIN	21	6	4.25	7	6	8.1	

Field	Description
SID	DELAY set identification number. See Remark 2. (Integer > 0)
NAME	Basic substructure name. (Character)
Pi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
Ti	Time delay, τ, for the designated grid/coordinate pair. (Real)

Remarks

1. Up to two dynamic load time delays may be defined on a single entry.
2. The DELAYS set identification number, SID, must be referenced on an RLOAD1 or RLOAD2 entry for frequency response, or a TLOAD1 or TLOAD2 entry for transient response in order to be used by NASTRAN-CORE.
3. Dynamic loads are applied in the substructure SOLVE operation.
4. Points referenced must exist in the SOLVEd structure.

DLOAD - *Dynamic Load Combination*

Description

Uses load set superposition to define a dynamic loading condition in frequency or transient response problems. Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries (for frequency response) or TLOAD1 or TLOAD2 entries (for transient response).

Format

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

Example

DLOAD	17	1.0	2.0	6	-2.0	7	2.0	8	
	-2.0	9							

Field	Description
SID	Load set identification number. (Integer > 0)
S	Scale factor. (Real)
Si	Scale factor on Li. (Real)
Li	Load set identification numbers. See Remark 2. (Integer > 0)

Remarks

1. The resultant load vector is given by:

$$\{P\} = S\sum_i S_i \cdot \{P_{Li}\}$$

2. The Li must be unique, and refer to RLOAD1 and/or RLOAD2 entries in frequency response analysis, or TLOAD1 and/or TLOAD2 entries in transient response analysis. Further, the SID must be unique across all Li and all other DLOAD entries, implying that one DLOAD entry cannot reference another.
3. Dynamic load sets must be selected in Case Control (DLOAD = SID) to be used by NASTRAN-CORE.
4. The DLOAD entry provides the mechanism for combining RLOAD1 and RLOAD2 load types in frequency response analysis, and TLOAD1 and TLOAD2 load types in transient analysis.

DMI - *Direct Matrix Input*

Description

Used to define matrix data blocks directly. Generates a matrix of the form:

$$[A] = \begin{bmatrix} A11 & A12 & \dots & \dots & \dots & A1n \\ A21 & A22 & \dots & \dots & \dots & A2n \\ . & . & & & & . \\ . & . & & & & . \\ . & . & & & & . \\ Am1 & \dots & \dots & \dots & \dots & Amn \end{bmatrix}$$

where the elements Aij may be real or complex single-precision or double precision numbers. In addition to the header entry, an additional DMI entry must be present for every non-null column of the matrix.

Format

The first entry is the matrix header:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	“0”	FORM	TIN	TOUT		M	N	

Subsequent DMI entries define nonnull columns:

DMI	NAME	J	I1	A(I1,J)	...	-etc.-	I2	A(I2,J)	
	...	-etc.-							

and so on, for each nonnull column

Example

Header record:

DMI	QQQ	0	2	3	3		4	2	

Column entries:

DMI	QQQ	1	1	1.0	2.0	3.0	4.0	3	
	5.0	6.0							
DMI	QQQ	2	2	6.0	7.0	4	8.0	9.0	

Field	Description
NAME	String that will be used in the DMAP sequence to reference the data block. (Character string, from one to eight characters in length, the first of which must be A-Z)
FORM	Matrix form. (Integer > 0) 1 Square matrix (unsymmetric). 2 General rectangular matrix. 6 Symmetric matrix.
TIN	Type of matrix being input: (Integer) 1 Real, single precision (one field is used per element) 2 Real, double precision (one field is used per element) 3 Complex, single precision (two fields are used per element) 4 Complex, double precision (two fields are used per element)

Field	Description
TOUT	Type of matrix that will be created: (Integer) 1 Real, single precision 3 Complex, single precision 2 Real, double precision 4 Complex, double precision
M	Number of rows in A. (Integer > 0)
N	Number of columns in A. (Integer > 0)
J	Column number of A. (Integer > 0)
Ii	Row number of A. (Integer > 0)
A(Ii,J)	Element of A (Real; See TIN field)

Remarks

1. The DMI matrix NAME must appear in a user-written DMAP, or alteration of a rigid format, in order to be used by NASTRAN-CORE. All of the rules governing the use of data blocks in DMAP sequences apply. In the example, the data block QQQ is defined to be the complex, single-precision rectangular 4x2 matrix:

$$[QQQ] = \begin{bmatrix} (1.0, 2.0) & (0.0, 0.0) \\ (3.0, 4.0) & (6.0, 7.0) \\ & \\ (5.0, 6.0) & (0.0, 0.0) \\ (0.0, 0.0) & (8.0, 9.0) \end{bmatrix}$$

2. A limit to the number of DMIs which may be defined is set by the size of the Data Pool Dictionary. The total number of DMIs may not exceed this size.
3. DMIAX names must be unique, and cannot duplicate names already appearing in the DMAP sequences (e.g., GEOM1, GEOM2, EPT, MPT, etc.)
4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices, the entire matrix must be input.
6. Only nonzero terms need be entered.
7. A blank field is not equivalent to a zero. If zero input is desired, the appropriate type of zero must be entered (that is, 0.0 or 0.0D0).
8. Complex input must have both the real and imaginary parts entered if either part is nonzero.
9. If A (IX,J) is followed by THRU in the next field and an integer row number IY after the THRU, then A (IX,J) will be repeated in each row through IY. The THRU must follow an element value. In the following example, 3.14 will appear in rows 3 through 6 of column 1, followed by a value of 2.0 in row 9.

DMI	QQQ	0	2	1	1		9	1	
DMI	QQQ	1	3	3.14	THRU	6	9	2.0	

DMIAX - *Axisymmetric Direct Matrix Input*

Description

Defines axisymmetric structure related direct input matrix terms.

Format

The first entry is the matrix header:

1	2	3	4	5	6	7	8	9	10
DMIAX	NAME	“0”	FORM	TIN	TOUT				

Subsequent DMIAX entry sets define nonnull columns:

DMIAX	NAME	GJ	CJ	NJ					
	GI	CI	NI	Xij	Yij				
	etc. for each nonzero row in the J-th column								

Example

DMIAX	B2PP	0	1	3	4				
DMIAX	B2PP	32							
	1027	3		4.35+6	2.27+3				

Field	Description
NAME	Name of matrix. (Character; one to eight characters, the first of which must be alphabetic)
FORM	Matrix form. (Integer > 0) 1 Square matrix (unsymmetric) 2 General rectangular matrix. 6 Symmetric matrix.
TIN	Type of matrix being input: 1 Real, single precision (one field is used per element.) 3 Complex, single precision (two fields are used per element.)
TOUT	Type of matrix that will be created: 1 Real, single precision 3 Complex, single precision 2 Real, double precision 4 Complex, double precision
GI, GJ	Grid, scalart, or extra point identification number. (Integer > 0; I, J indicates row, column order)
CI, CJ	Component number. (0 < Integer ≤ 6 if grid point, blank or zero if scalar or extra point. I, J indicates row, column order)
NI, NJ	Not used
Xij, Yij	Real and imaginary parts of matrix element; (GI, CI, NI) row and (GJ, CJ, NJ) column.

Remarks

1. This entry is allowed only if an AXIF entry is also present.
2. Matrices defined on DMIAX entries can be directly included in dynamics problems using the Case Control commands K2PP, B2PP, or M2PP for $[K^2_{pp}]$, $[B^2_{pp}]$, or $[M^2_{pp}]$ - type matrices, respectively, and K2GG, B2GG, or M2GG for $[K^2_{gg}]$, $[B^2_{gg}]$, or $[M^2_{gg}]$ - type matrices. The “NAME” used in Case Control is the name assigned to the matrix in field 2 of the header entry. In non-dynamics problems, the user must write DMAP commands that reference the matrix.
3. In addition to the header entry containing FORM, TIN, and TOUT, a logical entry consisting of two or more lines of input is needed for each nonnull column of the matrix.
4. If TIN = 1, the Yij’s must be blank.
5. Field 3 of the header entry must contain an integer 0.
6. For symmetric matrices, the entire matrix must be input.
7. Only nonzero terms need be entered.
8. DMIAX names must be unique, and cannot duplicate names already appearing in the DMAP sequences (e.g., GEOM1, GEOM2, EPT, MPT, etc.)

DMIG - *Direct Matrix Input at Grid Points*

Description

Defines structure-related direct input matrices.

Format

The first entry is the matrix header:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	FORM	TIN	TOUT			NCOL	

Subsequent DMIG entry sets define nonnull columns:

DMIG	NAME	GJ	CJ		GI	CI	Xij	Yij	
	GI	CI	Xij	Yij	-etc.-				

Example

DMIG	STIF	0	1	3	4				
DMIG	STIF	27	1		2	3	3.+5	3.+3	
	2	3	2.54+3	1.3+5	2	5	0.0	1.4	

Field	Description
NAME	Name of matrix. (Character; one to eight characters, the first of which must be alphabetic)
FORM	Matrix form. (Integer > 0) 1 Square matrix (unsymmetric) 2 General rectangular matrix 6 Symmetric matrix
TIN	Type of matrix being input: 1 Real, single precision (one field is used per element) 2 Real, double precision (one field is used per element) 3 Complex, single precision (two fields are used per element) 4 Complex double precision (two fields are used per element)
TOUT	Type of matrix that will be created: 1 Real, single precision 3 Complex, single precision 2 Real, double precision 4 Complex, double precision
NCOL	Number of columns in the input matrix (Required if FORM = 2 or 9, not used otherwise)
GI, GJ	Grid, scalar or extra point identification number. (Integer > 0; I, J indicates row, column order)
CI, CJ	Component number. (0 < Integer ≤ 6 if grid point, blank or zero if scalar or extra point. I, J indicates row, column order.)
Xij, Yij	Real and imaginary parts of matrix element.

Remarks

1. Matrices defined on DMIG entries can be directly included in dynamics problems using the Case Control commands K2PP, B2PP, or M2PP for $[K^2_{pp}]$, $[B^2_{pp}]$, or $[M^2_{pp}]$ - type matrices, respectively, and K2GG, B2GG, or M2GG for $[K^2_{gg}]$, $[B^2_{gg}]$, or $[M^2_{gg}]$ - type matrices. The “NAME” used in Case Control is the name assigned to the matrix in field 2 of the header entry. A matrix defined using DMIG entries may be used in the program by using the MTRXIN module.
2. In addition to the header entry containing FORM, TIN, and TOUT, a logical entry consisting of two or more lines of input is needed for each nonnull column of the matrix.
3. If TIN = 1, the Yij’s must be blank.
4. Field 3 of the header entry must contain an integer 0.
5. Only nonzero terms need be entered.
6. DMIG names must be unique, and cannot duplicate names already appearing in the DMAP sequences (e.g., GEOM1, GEOM2, EPT, MPT, etc.)
7. For symmetric matrices (IFO = 6), a given off-diagonal element may be input either above or below the diagonal. If both are specified the coefficient must be the same. If a full matrix is given the matrix must satisfy symmetry conditions; otherwise a fatal error will be issued.
8. DMIG entries can be created using the MATGPR module.

DPHASE - *Dynamic Load Phase Lead*

Description

Used in conjunction with the RLOAD1 and RLOAD2 entries to define the phase lead term, θ, in the frequency response loading equations.

Format

1	2	3	4	5	6	7	8	9	10
DPHASE	SID	P1	C1	THETA1	P2	C2	THETA2		

Example

DPHASE	4	21	6	2.1	8	6	7.2		

Field	Description
SID	DPHASE set identification number. (Integer > 0)
Pi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
THETAi	Phase lead, θ (in degrees), for designated grid/coordinate pair. (Real)

Remarks

1. Up to two dynamic load phase lead terms may be defined on a single entry.
2. The SID must be referenced on an RLOAD1 or RLOAD2 entry. See these entries for the manner in which θ is used.
3. Note that DPHASE only defines a phase term at Pi, Ci. The corresponding physical load is provided by DAREA reference on RLOAD1/RLOAD2 or TLOAD1/TLOAD2 entries.

DPHASES - *Dynamic Load Phase Lead, Substructure Analysis*

Description

Used in conjunction with the RLOAD1 and RLOAD2 entries to define the phase lead term, θ, in the frequency response loading equations.

Format

1	2	3	4	5	6	7	8	9	10
DPHASES	SID	NAME	P1	C1	THETA1	P2	C2	THETA2	

Example

DPHASES	4	SKIN	21	6	2.1	8	6	7.2	

Field	Description
SID	DPHASES set identification number. (Integer > 0)
NAME	Basic substructure name. (Character)
Pi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
THETAi	Phase lead, θ (in degrees), for designated grid/coordinate pair. (Real)

Remarks

1. Up to two dynamic load phase lead terms may be defined on a single entry.
2. The DPHASES entry is used in substructure SOLVE operation.
3. Points referenced must exist in the SOLVEd structure.

DSFACT - *Differential Stiffness Factor*

Description

Used to define a scale factor for applied loads and stiffness matrix in a normal modes with differential stiffness analysis.

Format

1	2	3	4	5	6	7	8	9	10
DSFACT	SID	B							

Example

DSFACT	97	-1.0							

Field	Description
SID	Unique set identification number. (Integer > 0)
B	Scale factor. (Real)

Remarks

1. Load sets must be selected in Case Control (DSCOEFF = SID) to be used by NASTRAN-CORE.
2. Fields four through ten must be blank.

DTI - *Direct Table Input*

Description

Used to define table data blocks directly.

Format

The first entry is the table header and trailer:

1	2	3	4	5	6	7	8	9	10
DTI	NAME	"0"	T1	T2	T3	T4	T5	T6	
	V1	V2	V3	-etc.-	"ENDREC"				

Subsequent DTI entry sets define additional table records:

DTI	NAME	IREC	V1	V2	V3	V4	V5	V6	
	V7	V8	-etc.-	"ENDREC"					

Example

DTI	XXX	0	3	4	4096	32768	1	0	
	7.8	400	XMPL	ENDREC					
DTI	XXX	1	2.0	-6	ABC	6.0D0	-1	2	
	4	-6.2	2.9	1	DEF	-1	ENDREC		

Field	Description
NAME	String that will be used in the DMAP sequence to reference the data block. (Character string, from one to eight characters in length, the first of which must be A-Z)
Ti	Trailer values. See Remark 3. ($0 \leq$ Integer ≤ 65535 or blank)
IREC	Record number. (Integer; record numbers are sequential beginning with 1)
Vi	Table values. (Any type; blank, integer, real, or character)
ENDREC	Character string indicating the end of a table record. See Remarks 2 and 5.

Remarks

1. The DTI matrix NAME must appear in a user-written DMAP, or alteration of a rigid format, in order to be used by NASTRAN-CORE. All of the rules governing the use of data blocks in DMAP sequences apply.
2. The continuation entry for the header/trailer record is optional. "ENDREC" is only required in this case, and follows the list of Vi values.
3. If trailer values are not specified, T1 = number of records, T2 through T6 = 0.
4. Continuation entries may be used to define records with as many terms as are necessary.
5. In addition to the header/trailer record, there must be one logical entry for each record in the table.
6. All fields following ENDREC must be blank.
7. DTI names must be unique, and cannot duplicate names already appearing in the DMAP sequences (e.g., POOL, UMF, GEOM1, GEOM2, EPT, etc.)

EIGB - *Buckling Analysis Data*

Description

Defines data for eigenvalue method.

Format

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	NEP	NDP	NDN	E	
	NORM	G	C						

Example

EIGB	13	FEER	0.1	2.5	2	1	1	0.0	
	MAX								

Field	Description
SID	Set identification number. (Unique integer > 0)
METHOD	Method of eigenvalue extraction. (Character string “LANC”, “INV”, “FEER”, “UINV”)

- LAN or LANC — Lanczos method, symmetric operations. For field entry descriptions, see EIGB,LAN
- INV — Inverse power method, symmetric matrix operations.
- FEER — Tridiagonal reduction method, symmetric matrix operations.
- UINV — Inverse power method, unsymmetric matrix operations.

Field	Description
L1, L2	Eigenvalue range of interest. See Remarks 3 and 4. (Real; 0.0 < L1 < L2)
NEP	Estimate of number of roots in positive range. Desired number of eigenvalues of smallest magnitude for METHOD = FEER (the default is automatically calculated to extract at least one accurate mode.) (Integer > 0)
NDP, NDN	Desired number of positive and negative roots. (Integer > 0; Ignored for METHOD=FEER. Default = 3*NEP)
E	Convergence criteria (optional.) (Real > 0.0)
NORM	Method for normalizing eigenvectors. (Character; either “MAX” or “POINT”)

- MAX — Normalize to unit value of the largest component in the analysis set.
- POINT — Normalize to unit value of the component defined in fields 3 and 4 (defaults to MAX if defined component is zero.)

Field	Description
G	Grid or scalar point identification number (only required if NORM = POINT.) (Integer > 0)
C	Component number (only required if NORM = POINT and G is a geometric grid point.) (Integer 1 through 6)

Remarks

1. Buckling analysis root extraction data sets must be selected in Case Control (METHOD = SID) to be used by NASTRAN-CORE.
2. The continuation entry is required.
3. The quantities L1 and L2 are dimensionless and specify a range in which the eigenvalues are to be found. An eigenvalue is a factor by which the pre-buckling state of stress (first subcase) is multiplied to produce buckling. If METHOD = FEER, L1 is ignored and L2 represents the maximum upper bound, in percent, on $\left|(\lambda_{FEER}/\lambda_{EXACT}) - 1\right|$ for acceptance of a computed eigensolution. Default = 0.1/n where n is the order of the stiffness matrix.
4. If NORM = MAX, components that are not in the analysis set may have values larger than unity.
5. If NORM = POINT, the selected component must be in the analysis set.

EIGB,LAN - *Buckling Analysis Data, Lanczos Method*

Description

Defines data for eigenvalue method.

Format

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	MSGLVL	ND	MAXSET	SHFSCL	

Example

EIGB	13	LAN	0.1	5.2		10			

Field	Description
SID	Set identification number. (Unique integer > 0)
METHOD	LAN or LANC
L1, L2	Eigenvalue range of interest. See Remarks 3 and 4. (Real)
MSGLVL	Diagnostic level. (Integer; 0 ≤ MSGLVL ≤ 4; Default = 0)
ND	Desired number of roots. (Integer > 0)
MAXSET	Diagnostic level. (Integer; 0 ≤ MSGLVL ≤ 4; Default = 0)
SHFSCL	Estimate of the first flexible mode natural frequency. (Real)

Remarks

1. Buckling analysis root extraction data sets must be selected in Case Control (METHOD = SID) to be used by NASTRAN-CORE.
2. The quantities L1 and L2 are dimensionless and specify a range in which the eigenvalues are to be found. An eigenvalue is a factor by which the pre-buckling state of stress (first subcase) is multiplied to produce buckling. computed
3. NORM = MAX, the only normalization method supported for Lanczos Method; components that are not in the analysis set may have values larger than unity.
4. The desired number of roots (ND) includes all roots including positive and negative roots
5. If L1<0.0, the negative eigenvalue range will be searched.
6. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
7. MSGLVL controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.
8. MAXSET is used to limit the maximum block size. It is otherwise set by the region size or by ND with a maximum size of 15. it may also be reset if there is not sufficient memory available. The default value is recommended.
9. If L1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If L1 is set to zero, negative eigenvalues are not calculated
10. A SHFSCL estimate may improve performance, especially when the first buckling mode occurs at high L values. If this field is blank, a value for SHFSCL is estimated automatically.
11. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. In this case, all roots and eigenvectors that pass the convergence checks are output, and the number is usually equal to or less than an integer multiple of MAXSET.

EIGC - *Complex Eigenvalue Extraction Data*

Description

Defines data needed to perform complex eigenvalue analysis.

Format

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E	ND0		
	ALPHAA1	OMEGAA1	ALPHAB1	OMEGAB1	L1	NE1	ND1		
	ALPHAA2	OMEGAA2	ALPHAB2	OMEGAB2	L2	NE2	ND2		

Example

EIGC	14	DET	POINTS	27		1.e-8			
	2.0	5.6	2.0	-3.4	2.0	4	4		
	-5.5	-5.5	5.6	5.6	1.5	6	3		

Field	Description
SID	Set identification number. (Integer > 0)
METHOD	Method of complex eigenvalue extraction. One of the character strings “INV”, “DET”, “HESS”, or “FEER”. • INV Inverse power method. • DET Determinant method. • HESS Upper Hessenberg method. • FEER Tridiagonal Reduction method
NORM	Method for normalizing eigenvectors, one of the character strings “MAX” or “POINT”. • MAX Normalize to a unit value for the real part and a zero value for the imaginary part, the component having the largest magnitude. • POINT Normalize to a unit value for the real part and a zero value for the imaginary part the component defined in fields 5 and 6–defaults to MAX if the magnitude of the defined component is zero.
G	Grid or scalar point identification number (required if and only if NORM = POINT.) (Integer > 0)
C	Component number (required if and only if NORM = POINT and G is a geometric grid point.) ($0 \leq$ Integer ≤ 6)
E	Convergence criterion (optional) For METHOD = FEER, error-tolerance on acceptable eigenvalues (default = .10/n, where n is the order of the stiffness matrix.) (Real $\geq$ 0.0)
(ALPHAAj, OMEGAAj), (ALPHABj, OMEGABj)	Two complex points defining a line in the complex plane. See Remarks 1 and 2. (Real)
Lj	Width of region in complex plane. (Real > 0.0 or blank for METHOD = FEER)
NEj	Estimated number of roots in each region. (Integer > 0; Ignored for METHOD = FEER)
NDj	Desired number of roots in each region. If METHOD = FEER, desired number of accurate roots (default is 1). (Integer > 0)

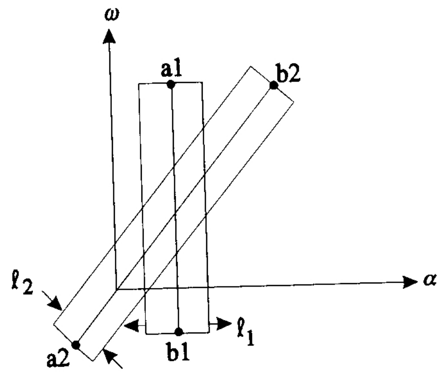


Figure 4-33 EIGC Search Regions

Remarks

1. Each continuation entry defines a rectangular search region. Any number of regions may be used and they may overlap. Roots in overlapping regions will not be extracted more than once.
2. For METHOD = FEER, ALPHAAj and OMEGAAj ($\alpha_{aj} + \omega_{aj} > 0$)define a circular search region centered at (α_{aj}, ω_{aj}) and of sufficient radius to encompass N_{dj} roots, If nonblank, the ALPHABj and OMEGABj fields are simply ignored.
3. Complex eigenvalue extraction data sets must be selected in Case Control (CMETHOD = SID) to be used.
4. The units of α, ω, and l (ALPHAij, OMEGAij, and Lj) are radians per unit time.
5. At least one continuation entry is required.
6. For the determinant method with no damping matrix, complex conjugates of the roots found are not printed.
7. See Section 10.4.4.5 of the Theoretical Manual for a discussion of convergence criteria.
8. For the Upper Hessenberg method, N_{dl} controls the number of eigenvectors computed. Only one continuation entry is considered and the (α,ω) pairs, along with the parameters l_1 and N_{el}, are ignored. Insufficient storage for HESS will cause the program to switch to INV.
9. The error tolerance, E, for the FEER method is with regard to

$$\frac{|\bar{p}_i - (\alpha_{aj}, \omega_{aj})|}{|p_i - (\alpha_{aj}, \omega_{aj})|} - 1 \qquad \text{for } [B] \neq [0]$$

and

$$\frac{|\bar{p}_i^2 - (\alpha_{aj}, \omega_{aj})^2|}{|p_i^2 - (\alpha_{aj}, \omega_{aj})^2|} - 1 \qquad \text{for } [B] = [0]$$

where $\bar{p}_i$ is a computed eigenvalue and p_i an exact eigenvalue.

10. The complex eigenvalue is given by $a + \omega = 2\pi f(i - 1/2g)$, where f is the frequency and g is the damping coefficient.
11. The default of NORM is MAX.

EIGP - *Poles in Complex Plane*

Description

Defines poles that are used in complex eigenvalue extraction.

Format

1	2	3	4	5	6	7	8	9	10
EIGP	SID	ALPHA1	OMEGA1	M1	ALPHA2	OMEGA2	M2		

Example:

EIGP	15	-5.2	0.0	2	6.3	5.5	3		

Field	Description
SID	Set identification number. See Remark 1. (Integer > 0)
ALPHAi, OMEGAi	Coordinates of point in complex plane. See Remark 2. (Real)
Mi	Multiplicity of complex root at pole defined by (ALPHAi, OMEGAi) (Integer > 0)

Remarks

1. The EIGP entry defines poles in the complex plane that are used with an EIGC entry having the same set identification number.
2. The units of ALPHAi, OMEGAi are radians per unit time.
3. Poles are used only in the determinant method.
4. Up to two poles may be defined on a single entry.

EIGR - *Real Eigenvalue Extraction Data*

Description

Defines data needed to perform real eigenvalue analysis.

Format

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	F1	F2	NE	ND	NZ	E	
	NORM	G	C						

Example

EIGR	13	INV	0.0	100.0	7	7	0		
	MAX								

Field	Description
SID	Set identification number. (Integer > 0)
METHOD	Method of eigenvalue extraction. One of the following character strings. • LAN or LANC — Lanczos method. For field entry descriptions, see EIGR, LAN • INV — Inverse power method, symmetric matrix operations. • GIV — Givens method of tridiagonalization. • MGIV — Modified Givens method (see Remark 11.) • HOU — Householder method of tridiagonalization • FEER — Tridiagonal reduction method, symmetric matrix operations. • FEER-Q — See Remark 12. • FEER-X — See Remark 12.
F1,F2	Frequency range of interest (required for METHOD = INV, or UINV). (Real ≥ 0.0; F1 ≤ F2) • If METHOD = GIV, frequency range over which eigenvectors are desired. The frequency range is ignored if ND > 0, in which case the eigenvectors for the first ND positive roots are found. Real, F1 ≤ F2. • If METHOD = FEER, F1is the center of range of interest. Default is F1 = 0.0, Real ≥ 0.0, and F2 is the acceptable relative error tolerance, as a percentage, on frequency-squared. Default, as a percentage, is 0.1/n where n is the order of the stiffness matrix.
NE	Estimate of number of roots in range (required for METHOD = INV, or UINV, ignored for METHOD = FEER and LANC.) For METHOD = GIVENS, number of roots to be printed, default all, rigid roots included. (Integer > 0)
ND	Desired number of roots for METHOD =INV or UNIV, (Default is 3 NE) (Integer > 0). Desired number of eigenvectors for METHOD = GIV (Integer > 0). Desired number of roots and eigenvectors for METHOD = FEER and LANC (default is automatically calculated to extract at least one accurate mode.)
E	Mass orthogonality test parameter. No test will be made if E = 0 (default = 0.0) (Real ≥ 0)

Field	**Description**
NORM	Method for normalizing eigenvectors. One of the character string MASS, MAX, or POINT. • MASS Normalize to unit value of the generalized mass. • MAX Normalize to unit value of the largest component in the analysis set. • POINT Normalize to unit value of the component defined in fields 3 and 4. Defaults to MAX if defined component is zero.
G	Grid or Scalar point ID. Required only if NORM=POINT. (Integer ≥ 0)
C	Component number. Required only if NORM = POINT and G is a geometric grid point. (Integer; any value in the range 1-6)

Remarks

1. Real eigenvalue extraction data sets must be selected in Case Control (METHOD = SID) to be used by NASTRAN-CORE.
2. The units of F1 and F2 are cycles per unit time. If METHOD = FEER, F2 represents the maximum upper bound, in percent, on:

$$\frac{\omega^2_{FEER}}{\omega^2_{EXACT}} - 1$$

   for acceptance of a computed eigensolution
3. The continuation entry is optional.
4. If METHOD = GIV or HOU, all eigenvalues are found.
5. If METHOD = GIV or HOU, the mass matrix for the analysis set must be positive definite.This means that all degrees of freedom, including rotations, must have mass properties. OMIT Bulk Data entries may be used to remove massless degrees of freedom.
6. A nonzero value of E in field 9 also modifies the convergence criteria.
7. If NORM = MAX, components that are not in the analysis set may have values larger than unity.
8. If NORM = POINT, the selected component must be in the analysis set.
9. If METHOD = GIV or HOU and rigid body modes are present, F1 should be set to zero if the rigid body eigenvectors are desired.
10. The desired number of roots (ND) includes all roots previously found, such as rigid body modes determined with the use of the SUPORT Bulk Data entry, or the number of roots previously checkpointed when restarting and APPENDing the eigenvector file. The APPEND feature is available in the case of the Determinant, Inverse Power and FEER methods of eigenvalue extraction.
11. Givens and Householder method require the mass matrix to not be singular. The MGIV method allows the mass matrix to be singular. However, the dynamic matrices could be bigger, or much bigger, which would require more CPU time and core space.
12. The rigid body frequencies are zero substituted unless FEER-X is requested. If FEER-Q is requested, certain key areas in FEER computations are done in quad precision (Real*16) for 32-bit word machines and in double precision for 60- and 64-bit word machines. The FEER-Q request would yield much better rigid body eigenvalues, but it may take two to three times longer to compute than FEER or FEER-X.

EIGR, LAN - *Real Eigenvalue Analysis Data, Lanczos Method*

Description

Defines data for eigenvalue method.

Format

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	F1	F2	MSGLVL	ND	MAXSET	SHFSCL	
	NORM								

Example

EIGR	13	LAN	0.1	5.2		10			
	MAX								

Field	Description
SID	Set identification number. (Unique integer > 0)
METHOD	LAN or LANC
F1, F2	Eigenvalue range of interest. See Remarks 3 and 4. (Real)
MSGLVL	Diagnostic level. (Integer; 0 ≤ MSGLVL ≤ 4; Default = 0)
ND	Desired number of roots. (Integer > 0)
MAXSET	Maximum number of vectors in block
SHFSCL	Estimate of the first flexible mode natural frequency. (Real)
NORM	Method for normalizing eigenvectors. One of the character string MASS or MAX. (Default=MASS) • MASS Normalize to unit value of the generalized mass. • MAX Normalize to unit value of the largest component in the analysis set.

Remarks

1. Real eigenvalue analysis extraction data sets must be selected in Case Control (METHOD = SID) to be used by NASTRAN-CORE.
2. The units of F1 and F2 are cycles per unit time.
3. If NORM = MAX, components that are not in the analysis set may have values larger than unity.
4. The desired number of roots (ND) includes all roots previously found, such as rigid body modes determined with the use of the SUPORT Bulk Data entry. The roots are found in order of increasing magnitude; that is those closest to zero are found first.
5. If F1<0.0, the negative eigenvalue range will be searched.
6. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
7. MSGLVL controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.
8. MAXSET is used to limit the maximum block size. It is otherwise set by the region size or by ND with a maximum size of 15. it may also be reset if there is not sufficient memory available. The default value is recommended.

9. If F1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If F1 is set to zero, negative eigenvalues are not calculated
10. A SHFSCL estimate may improve performance, especially when large mass techniques are used in enforced motion analysis. Large mass techniques can cause a large gap between the rigid body and flexible frequencies. If this field is blank, a value for SHFSCL is estimated automatically.
11. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. In this case, all roots and eigenvectors that pass the convergence checks are output, and the number is usually equal to or less than an integer multiple of MAXSET.

EIGRL - *Real Eigenvalue Extraction Data, Lanczos Method*

Description

Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Format

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	F1	F2	ND	MSGLVL	MAXSET	SHFSCL	NORM	

Example

EIGRL	13	0.1	5.2	10					

Field	Description
SID	Set identification number
F1,F2	Frequency range of interest. (Real ≥ 0.0)
ND	Desired number of roots. (Integer > 0)
MSGLVL	Diagnostic level. (Integer; 0 ≤ MSGLVL ≤ 4; Default = 0)
MAXSET	Maximum number of vectors in block.
SHFSCL	Estimate of the first flexible mode natural frequency. (Real)
NORM	Method for normalizing eigenvectors, one of the character values MASS, or MAX. (Character; Default = “MASS”) • MASS Normalize to unit value of the generalized mass. • MAX Normalize to unit value of the largest component in the analysis set.

Remarks

1. Real eigenvalue extraction data sets are selected using the Case Control command METHOD = SID.
2. The units of F1 and F2 are cycles per unit time in vibration analysis. These values are dimensionless eigenvalues in buckling analysis.
3. If NORM = MAX, components that are not in the analysis set may have values larger than unity.
4. The desired number of roots (ND) includes all roots previously found, such as rigid body modes determined with the use of the SUPORT Bulk Data entry. The roots are found in order of increasing magnitude; that is those closest to zero are found first.
5. In vibration analysis, if F1<0.0, the negative eigenvalue range will be searched. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
6. MSGLVL controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.
7. MAXSET is used to limit the maximum block size. It is otherwise set by the region size or by ND with a maximum size of 15. it may also be reset if there is not sufficient memory available. The default value is recommended.
8. If F1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If F1 is set to zero, negative eigenvalues are not calculated

9. A SHFSCL estimate may improve performance, especially when large mass techniques are used in enforced motion analysis. Large mass techniques can cause a large gap between the rigid body and flexible frequencies. If this field is blank, a value for SHFSCL is estimated automatically.
10. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. In this case, all roots and eigenvectors that pass the convergence checks are output, and the number is usually equal to or less than an integer multiple of MAXSET.

ENDDATA - *End of Bulk Data*

Description

Defines the end of the Bulk Data section.

Format

1	2	3	4	5	6	7	8	9	10
ENDDATA									

Alternate Format 1

ENDATA									

Alternate Format 2

END DATA									

Remarks

1. This entry is required even if Bulk Data entries do not exist in the input file.
2. ENDDATA may begin in columns 1 or 2. If the first alternate format is used, ENDATA may begin in columns 1, 2, or 3. If the second alternate form is used, END DATA must begin in column 1.
3. A fatal error will result if this entry is not present.
4. Extraneous entries may follow this entry in the input file except when INPUT module data follows or when the UMF entry “FINIS” follows.

EPOINT - *Extra Point*

Description

Defines extra points for use in dynamics problems.

Format

1	2	3	4	5	6	7	8	9	10
EPOINT	ID1	ID2	ID3	-etc.-					

Example:

EPOINT	3	18	1	4	16	2			

Alternate Format and Example:

EPOINT	ID1	"THRU"	ID2						
EPOINT	17	THRU	43						

Field	Description
IDi	Extra point identification number. (Integer > 0)

Remarks

1. Extra point identification numbers must be unique with respect to all other structural and scalar points.
2. Coordinates to be used in transfer functions are defined on this entry. (See the TF entry for dynamic transfer function definition.)
3. The alternate format defines a set of extra points, ID1 through ID2, inclusive. ID1 must be less than ID2.

FORCE - *Static Load*

Description

Defines a static load at a grid point.

Format

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Example

FORCE	2	6	6	2.93	0.0	1.0	0.0		

Field	Description
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 or blank; Default = 0)
F	Scale factor. (Real)
Ni	Vector components in the CID coordinate system. (Real)

Remarks

1. The static load applied at grid point G is given by

$$\vec{f} = F\vec{N}$$

where $\vec{N}$ is the vector whose components are defined in fields 6, 7, and 8, and

$$|\vec{f}| = F \cdot |\vec{N}|$$

2. A CID of zero or blank references the basic coordinate system.
3. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
4. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.

FORCE1 - *Static Load*

Description

Defines a static load at a grid point by specification of a magnitude and two grid points which determine the direction.

Format

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

Example

FORCE1	6	13	-2.93	16	13				

Field	Description
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of force. (Real)
Gi	Grid point identification numbers. (Integer > 0; G1 and G2 non-coincident)

Remarks

1. The load vector defined by this entry is applied at grid point G, and is given by:

$$\vec{f} = F \cdot \vec{N}$$

where $\vec{N}$ is a unit vector in the direction of G1 to G2, or

$$\vec{N} = \frac{\overrightarrow{G2} - \overrightarrow{G1}}{\left|\overrightarrow{G2} - \overrightarrow{G1}\right|}$$

2. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
3. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.

FORCE2 - *Static Load*

Description

Defines a static load by specification of a magnitude and four grid points which determine the direction.

Format

1	2	3	4	5	6	7	8	9	10
FORCE2	SID	G	F	G1	G2	G3	G4		

Example

FORCE2	6	13	-2.93	16	13	17	13		

Field	Description
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of force. (Real)
Gi	Grid point identification numbers. (Integer > 0; G1 and G2 cannot be coincident, nor can G3 and G4.)

Remarks

1. The load vector defined by this entry is applied at grid point G, and is given by:

$$\vec{f} = F \cdot \vec{N}$$

   where $\vec{N}$ is a unit vector in the direction of the cross product of the vectors G1 to G2, and G3 to G4. The cross product must not be zero.

2. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
3. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.

FORCEAX - *Axisymmetric Static Load*

Description

Defines a static loading for a model containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
FORCEAX	SID	RID	HID	S	FR	FP	FZ		

Example

FORCEAX	1	2	3	2.0	0.1	0.2	0.3		

Field	Description
SID	Load set identification number. (Integer > 0)
RID	Axisymmetric ring (RINGAX) identification number. (Integer > 0)
HID	Harmonic identification number(s). Multiple harmonics can be entered as a sequence; see Remark 4. (Integer ≥ 0, or mixed character/integer string)
S	Scale factor for load. (Real)
FR, FP, FZ	Load components in r, ϕ, z directions. (Real)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Axisymmetric loads must be selected in Case Control (LOAD = SID) to be used by NASTRAN-CORE.
3. A separate entry is needed for the definition of the force associated with each harmonic.
4. A sequence of harmonics can be entered in the HID field using the form, “Sn1Tn2” where n1 is the start of the sequence and n2 is the end of the sequence. For example, “S0T10” indicates loading for harmonics 0 through 10.
5. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
6. For a discussion of the axisymmetric solid problem see Section 5.11 of the Theoretical Manual.

FREQ - *Frequency List Definition*

Description

Defines a set of frequencies to be used in the solution of frequency response problems.

Format

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	-etc.-						

Example

FREQ	13	0.0	1.0	2.0	2.2	2.4	2.45	2.46	
	2.47	2.48	2.52	2.6	3.				

Field	Description
SID	Frequency set identification number. (Integer > 0)
Fi	Frequency value. (Real ≥ 0.0)

Remarks

1. Frequencies are expressed in units of cycles per unit time.
2. Frequency sets must be selected in Case Control with FREQ = SID
3. FREQ, FREQ1, FREQ3, FREQ4, and FREQ5 entries with the same setid are combined to create the set.

Note: NOTE: FREQ3, FREQ4, and FREQ5 entries only work in modal frequency response.

4. Frequencies in a set are considered to be duplicates if

$$|F_i - F_{i-1}| < DFREQ \times |F_{max} - F_{min}|$$

where DFREQ is a parameter (default = 10^{-5}), F_{max} is the highest frequency in the set, and F_{min} is the lowest frequency in the set. Duplicates will be removed from the set.

FREQ1 - *Frequency List, definition, Uniform Increments*

Description

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and number of increments desired.

Format

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

Example

FREQ1	6	2.9	0.5	13					

Field	Description
SID	Frequency set identification number. (Integer > 0)
F1	Starting frequency. (Real ≥ 0.0)
DF	Frequency increment. (Real ≥ 0.0)
NDF	Number of frequency increments. (Integer > 0)

Remarks

1. The units for the frequency F1 and the frequency increment DF are cycles per unit time.
2. The frequencies defined by this entry are given by:

$$F_{i+1} = F_1 + i \cdot DF \quad ; \quad \mathrm{i} = 0,1,\ldots,NDF$$

3. Frequency sets must be selected in Case Control with FREQ = SID
4. FREQ, FREQ1, FREQ3, FREQ4, and FREQ5 entries with the same setid are combined to create the set.

Note: NOTE: FREQ3, FREQ4, and FREQ5 entries only work in modal frequency response.

5. Frequencies in a set are considered to be duplicates if

$$|F_i - F_{i-1}| < DFREQ \times |F_{max} - F_{min}|$$

where DFREQ is a parameter (default = 10^{-5}), F_{max} is the highest frequency in the set, and F_{min} is the lowest frequency in the set. Duplicates will be removed from the set.

FREQ2 - *Frequency List, Logarithmic Increments*

Description

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, ending frequency, and number of logarithmic increments desired.

Format

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

Example

FREQ2	21	1.0	8.0	6					

Field	Description
SID	Frequency set identification number. (Integer > 0)
F1	Starting frequency. (Real > 0.0)
F2	Ending frequency. (Real > 0.0; F2 > F1)
NF	Number of logarithmic intervals. (Integer > 0)

Remarks

1. Frequencies F1 and F2 are expressed in units of cycles per unit time.
2. The frequencies defined by this entry are given by:

$$F_{i+1} = F_1 e^{i \cdot d} \quad ; \quad \text{i} = 0,1,\ldots, NF$$

where

$$d = \frac{1}{NF} \ln\left(\frac{F_2}{F_1}\right)$$

In the example shown, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569, and 8.0 cycles per unit time.

3. Frequency sets must be selected in Case Control with FREQ = SID.
4. FREQ, FREQ1, FREQ3, FREQ4, and FREQ5 entries with the same setid are combined to create the set.

Note: NOTE: FREQ3, FREQ4, and FREQ5 entries only work in modal frequency response.

---

5. Frequencies in a set are considered to be duplicates if

$$|F_i - F_{i-1}| < DFREQ \times |F_{max} - F_{min}|$$

where DFREQ is a parameter (default = 10^{-5}), F_{max} is the highest frequency in the set, and F_{min} is the lowest frequency in the set. Duplicates will be removed from the set.

FREQ3 - *Frequency List defined by a formula*

Description

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, ending frequency, and values for use in an equation.

Format

1	2	3	4	5	6	7	8	9	10
FREQ3	SID	F1	F2	TYPE	NEF	BIAS			

Example

FREQ3	21	1.0	80.0		11	1.0			

Field	Description
SID	Frequency set identification number. (Integer > 0)
F1	Starting frequency. (Real > 0.0)
F2	Ending frequency. (Real > 0.0; F2 > F1)
TYPE	Interpolation metiod (BCD, LINEAR or LOG, default = LINEAR)
NEF	Number of excitation frequencies to define using each set of modes in the region. (Integer > 0)
BIAS	Value used in the equation to determine how the frequencies are distributed. See Remark 6 (Real, > 0.0, default = 1.0)

Remarks

1. FREQ3 entries define loading frequencies based on the natural frequencies of your model and are only used in modal frequency response.
2. Frequencies F1 and F2 are expressed in units of cycles per unit time.
3. Frequency sets must be selected in Case Control with FREQ = SID.
4. FREQ, FREQ1, FREQ3, FREQ4, and FREQ5 entries with the same setid are combined to create the set.
5. Frequencies in a set are considered to be duplicates if

$$|F_i - F_{i-1}| < DFREQ \times |F_{max} - F_{min}|$$

   where DFREQ is a parameter (default = 10^{-5}), F_{max} is the highest frequency in the set, and F_{min} is the lowest frequency in the set.

   Duplicates will be removed from the set.

6. BIAS is used to determine the distribution of the frequencies between the natural frequencies in the range. BIAS = 1.0 gives a uniform spacing, BIAS < 1.0 causes concentration in the mid-point between natural frequencies and BIAS > 1.0 causes concentrations near the natural frequencies.
7. For each set of modes which fall in the range, the following equation is used to determine the frequencies in the range between F1 and F2. .

$$F_i = \frac{1}{2}(f1 + f2) + \frac{1}{2}(f2 - f1)|\xi_i|^{\frac{1}{BIAS}} \times SIGN(\xi_i)$$

where:

The value of ξ_i (the location coordinate) is defined by

$$\xi_i = -1 + \frac{2i-1}{NEF-1}$$

i = varies from 1 to NEF

f1 = lower natural frequency in the pair being used (or F1 for the first mode found in the range F1-F2)

f2 = highest natural frequency in the pair being used (or F1 for the highest mode in the range F1-F2

Fi = is the i-th exicitation frequency (or logarithm of it if LOG is specified)

f = the frequency (or logarithm of the frequency if LOG is specified)

FREQ4 - *Frequency List defined by spread around natural frequencies*

Description

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, ending frequency and a spread factor.

Format

1	2	3	4	5	6	7	8	9	10
FREQ4	SID	F1	F2	FSPD	NFM				

Example

FREQ4	21	1.0	200.0	.01	7				

Field	Description
SID	Frequency set identification number. (Integer > 0)
F1	Starting frequency. (Real > 0.0)
F2	Ending frequency. (Real > 0.0; F2 > F1)
FSPD	Frequency spread - fraction used to determine the range around the natural frequencies (1.0>Real>0.0, default = 0.1)
NFM	Number of excitation frequencies to define using each mode in the region. (Integer, odd > 0, default = 3) - if an even number is entered, NFM + 1 will be used

Remarks

1. FREQ4 entries define loading frequencies based on the natural frequencies of your model and are only used in modal frequency response.
2. Frequencies F1 and F2 are expressed in units of cycles per unit time.
3. FREQ, FREQ1, FREQ3, FREQ4, and FREQ5 entries with the same setid are combined to create the set.
4. For each natural frequency in the range, the frequencies in the set are defined by

$$F_i = F_n(1 - FSPD) + \frac{2 \times FSPD}{(NFM - 1)} \times (i - 1)$$

where:

F_i = i^{th} loading frequency in the set - cycles per unit time (values outside the range will be ignored)

i = counter, ranging from 1 to NFM for each mode in the range

F_n = natural frequency of the n^{th} mode in the range - cycles per unit time

5. Frequencies in a set are considered to be duplicates if

$$|F_i - F_{i-1}| < DFREQ \times |F_{max} - F_{min}|$$

where DFREQ is a parameter (default = 10^{-5}), F_{max} is the highest frequency in the set, and F_{min} is the lowest frequency in the set. Duplicates will be removed from the set.

FREQ5 - *Frequency List defined as fractions of natural frequencies*

Description

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, ending frequency and scaling factors.

Format

1	2	3	4	5	6	7	8	9	10
FREQ5	SID	F1	F2	FRAC1	FRAC2	FRAC3	FRAC4	FRAC5	
	FRAC6	FRAC7	FRAC8	FRAC9	-etc-				

Example

FREQ5	21	1.0	200.0	.91	.95	1.0	1.05		

Field	Description
SID	Frequency set identification number. (Integer > 0)
F1	Starting frequency. (Real > 0.0)
F2	Ending frequency. (Real > 0.0; F2 > F1)
$FRAC_i$	Scaling factors - fraction used to determine the loading frequencies (Real>0.0)

Remarks

1. FREQ5 entries define loading frequencies based on the natural frequencies of your model and are only used in modal frequency response.
2. Frequencies F1 and F2 are expressed in units of cycles per unit time.
3. FREQ, FREQ1, FREQ3, FREQ4, and FREQ5 entries with the same setid are combined to create the set.
4. For each natural frequency in the range, the frequencies in the set are defined by

$$F_i = FRAC_i \times F_n$$

   where:

   F_i = i^{th} loading frequency in the set - cycles per unit time (values outside the range will be ignored)

   i = counter, ranging from 1 to number of values provided for $FRAC_i$

   F_n = natural frequency of the n^{th} mode in the range - cycles per unit time

5. Frequencies in a set are considered to be duplicates if

$$|F_i - F_{i-1}| < DFREQ \times |F_{max} - F_{min}|$$

   where DFREQ is a parameter (default = 10^{-5}), F_{max} is the highest frequency in the set, and F_{min} is the lowest frequency in the set. Duplicates will be removed from the set.

GAP - *Linear GAP*

Description

Defines a GAP - may be used to measure gap openings and/or apply gaps.

Format

1	2	3	4	5	6	7	8	9	10
GAP	GAPID	G1	G2	OPENING	CID	DOF	STATUS	SET	
	S	OCID	S1	S2	S3	SPOPEN	TOL		

Example

GAP	2	6	7	0.05					

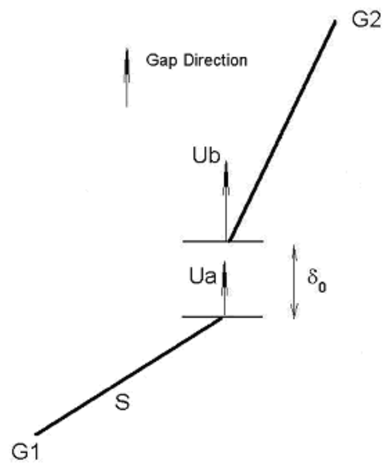
Field	Description
GAPID	ID of SPOINT which will be used to measure the GAP opening
G1	Starting Grid point identification number. (Integer > 0)
G2	Ending Grid point identification number. (Integer > 0 or blank)
OPENING	Initial GAP opening. (Real ≥ 0.0 or blank)
CID	Coordinate system identification number. (Integer ≥ 0 or blank)
DOF	Degree of Freedom (Integer >0 or blank)
STATUS	Flag to indicate initial status of GAP for interative solution (Integer, default = 0)
SET	Flag to indicate if the GAP DOF is to be placed in a specific set (is to be constrained or not (BCD or Integer - See Remark 9
S	Fractional location of the GAP on the line connecting grids GA and GB (Real, default = 0.0)
OCID	Coordinate system used to define the GAP location relative to G1. (Integer or blank)
S1,S2,S3	Offset vector (In OCID) of GAP from GA if OCID is provided (Real or blank)
SPOPEN	SPOINT id to be used to represent the initial opening (Integer >0 or blank)
TOL	Tolerance used to check for coincidence of G1 and G2 (Real, default = 1.E-6)

Remarks

1. The GAP has several purposes. It may be used to model a GAP, measure a GAP, or to impose a GAP.
2. In solutions 1 and 2 (linear statics and inertia relief), GAP entries which are not constrained will be constrained to have positive displacements only. This involves an iterative solution using DMAP module LINGAP In this case, the GAP models a GAP between two GRID points (forbidding penetration) or a GAP between a GRID point and ground.
3. In other solutions, if the GAPID (SPOINT) is not constrained, the GAP will provide a measure of the GAP between two GRID points or a single GRID point and a reference location. In transient response, NOLIN's may be used to apply loads based on the GAP opening. A force applied on the GAP dof will be distributed by the GAP to the connecting point(s). In this way, a GAP may be used to model a GAP between two GRID points, or between a single GRID point and ground in transient response. NOTE: the initial opening is set to 0.0 in dynamic solutions (non-zero initial openings in dynamics will be a future addition).
4. If the GAPID is constrained, the GAP element enforces a GAP between two GRID points or a single GRID point and ground..
5. The GAPID is the id of an SPOINT (automatically created if an SPOINT entry with this id is not defined) which will be used to represent (measure) the GAP opening.

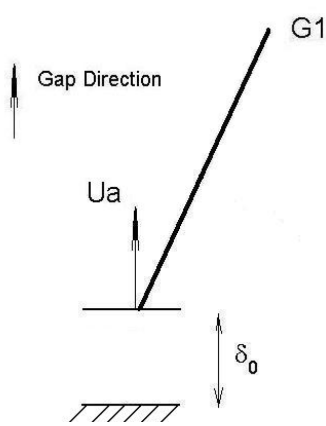
6. SPOPEN is a SPOINT id (automatically created if an SPOINT entry with this id is not defined) which is used to apply the initial GAP opening (OPENING). If a value is provided for OPENING and SPOPEN is not provided, an SPOINT will automatically be created. The automatically created SPOINT ids will start with the value of SYSTEM cell 132. Any SPOINT used to represent the initial opening is automatically constrained to have the initial opening as a displacement (in statics, otherwise to 0.0 in dynamics).
7. If both G1 and G2 are provided, the gap opening increases with positive motion of internal point B and decreases with positive motion of internal point A in the gap direction.

$$\delta = U_b - U_a + \delta_0$$



8. If both G1 and G2 are provided, and the distance between them is less than TOL, then CID and/or DOF must be specified.
9. If only G1 is provided, the gap opening increases with positive motion of internal point A. This is done for the iterative GAP.For this case, the GAP opening is calculated as

$$\delta = Ua + \delta_0$$



10. If G1 and G2 are provided, but CID is blank, the gap direction will be along the line from G1 to G2. In this case, if the initial GAP opening (OPENING) is not provided, the distance from G1 to G2 will be used.
11. If G2 is not provided and CID is not provided, the GAP direction is defined in the BASIC coordinate system. If DOF is not provided in this case, dof 1 is assumed.

12. SET is used to determine if the GAP dof is to go into a specified dof set. The default value indicates that no special action is to be done on declaring the set for the GAPID (Except in SOL's 1 and 2, where GAPID dof are automatically treated as if they are on A-set entries unless SET is 'S' or 2, in which case, they are placed in the S-set- this option can be overridden by setting PARAM,SOLTYP,0 in SOL 1 or 2). If desired, GAPID dof can be placed in the A-set or S-set by using this field. Placing "A" in this field is equivalent to entering the GAPID on an ASET entry, in that it places the GAPID in the A-set and forces a reduction. Placing "S" in this field constrains the GAPID and non-zero constraints can then be declared using SPC entries. An integer value of 1 is equivalent to "A" and an integer value of 2 is equivalent to "S".
13. A non-zero value for STATUS indicates an "assumed" closed initial state in SOL's 1 and 2. It has no meaning in any other solution.

GENEL - *General Element*

Description

Defines a general element using either of two approaches as follows.

1. The stiffness approach:

$$\begin{Bmatrix} f_1 \\ \hline f_d \end{Bmatrix} = \begin{bmatrix} K & -KS \\ \hline -S^T K & S^T KS \end{bmatrix} \begin{Bmatrix} u_i \\ \hline u_d \end{Bmatrix} \text{ ,or}$$

2. The flexibility approach:

$$\begin{Bmatrix} u_i \\ \hline f_d \end{Bmatrix} = \begin{bmatrix} Z & S \\ \hline -S^T & 0 \end{bmatrix} \begin{Bmatrix} f_i \\ \hline u_d \end{Bmatrix} \text{ , where}$$

$$u_i = [u_i 1, u_i 2, \ldots, u_{im}]^T \quad ,$$

$$u_d = [u_d 1, u_d 2, \ldots, u_{dn}]^T \quad ,$$

$$[KZ]^T = [KZ] = [K] \text{ or } [Z] = \begin{bmatrix} KZ_{11} & KZ_{12} & \ldots & KZ_{lm} \\ . & KZ_{22} & \ldots & \ldots \\ . & . & . & . \\ . & . & . & . \\ KZ_m 1 & \ldots & \ldots & KZ_{mn} \end{bmatrix}$$

$$[S] = \begin{bmatrix} S_{11} & \ldots & \ldots & \ldots & S_{1n} \\ . & & & & . \\ . & & & & . \\ . & & & & . \\ S_m 1 & \ldots & \ldots & \ldots & S_{mn} \end{bmatrix}$$

The required input is the {ui} list and the lower triangular portion of [K] or [Z]. Additional input may include the {ud} list and [S]. If [S] is input, {ud} must also be input. If {ud} is input but [S] is omitted, [S] is internally calculated. In this case, {ud} must have six and only six degrees of freedom. If [S] is not required, both {ud} and [S] are omitted.

Format

1	2	3	4	5	6	7	8	9	10
GENEL	EID		UI1	CI1	UI2	CI2	UI3	CI3	
	UI4	CI4	UI5	CI5	UI6	CI6	UI7	CI7	
				etc.					
	UIm - The last item in the UI-list appears in one of fields 2, 4, 6, 8.								
	"UD"		UD1	CD1	UD2	CD2	UD3	CD3	
				etc.					
	UDn - The last item in the UD list appears in one of fields 2, 4, 6, or 8.								

1	2	3	4	5	6	7	8	9	10
	“K” or “Z”	KZ11	KZ21	KZ31	etc.		KZ22	KZ32	
	etc.		KZ33	KZ43	etc.				
				etc.					
	KZm - The last item in the K or Z matrix appears in one of fields 2 through 9.								
	“S”	S11	S12	etc.		S21	etc.		
	Smn - The last item in the S matrix appears in one of fields 2 through 9.								

Field	Description	Type
EID	Unique element identification number, a positive integer.	Integer > 0
UI1, CI1 etc.; UD1, ED1, etc.	Identification numbers of coordinates in the UI or UD list, in sequence corresponding to the [K], [Z], and [S] matrices. Ui and UDi are grid point numbers, and CIi and CDi are the component numbers. If a scalar point is given, the component number is zero.	
KZij	Values of the [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list.	
Sij	Values of the [S] matrix ordered by rows, according to the UD list.	
UD, K, Z, S	Character strings indicating the start of data belonging to UD, [K], [Z], or [S].	

Remarks

1. When the stiffness matrix, K, is input, the number of significant digits should be the same for all terms.
2. Double-field format may be used for input of K or Z.

Example

Let element 629 be defined by

$$\{u_i\} = [1-1, 13-4, 42, 24-2]^T,$$

$$\{u_d\} = [6-2, 33]^T,$$

where i-j means the jth component of grid point i. Points 42 and 33 are scalar points.

$$[K] = \begin{bmatrix} 1.0 & 2.0 & 3.0 & 4.0 \\ 2.0 & 5.0 & 6.0 & 7.0 \\ 3.0 & 6.0 & 8.0 & 9.0 \\ 4.0 & 7.0 & 9.0 & 0.0 \end{bmatrix}, \qquad [S] = \begin{bmatrix} 1.5 & 2.5 \\ 3.5 & 4.5 \\ 5.5 & 6.5 \\ 7.5 & 8.5 \end{bmatrix}$$

The data entries necessary to input this general element are shown below:

1	2	3	4	5	6	7	8	9	10
GENEL	629		1	1	13	4	42	0	
	24	2							
	UD		6	2	33	0			
	K	1.0	2.0	3.0	4.0	5.0	6.0	7.0	

1	2	3	4	5	6	7	8	9	10
	8.0	9.0	0.0						
	S	1.5	2.5	3.5	4.5	5.5	6.5	7.5	
	8.5								

GEOMCHK - Element Geometry checking control

Description

Allows you to set the criteria used in performing geometry checks on the elements.

Format

1	2	3	4	5	6	7	8	9	10
GEOMCHK	ELNAME	SKEW	TAPER	WARP	MINANGLE	MAXANGLE	ARATIO	EDGERAT	
	BETA	OFFSET							

Example

GEOMCHK	HEXA			20.			10.		

Field	Description
ELNAME	Element name - see remark 2 - (BCD - no default)
SKEW	Allowable Skew Angle of a face in degrees (Real > 0.0 see Remark 4 for defaults)
TAPER	Allowable taper ratio on a face (Real > 0.0 see Remark 4 for defaults)
WARP	Maximum angle (in degrees) between the normals of two triangles defining a face (Real > 0.0 see Remark 4 for defaults)
MINANGLE	Minimum interior angle in degrees (Real > 0.0 see Remark 4 for defaults)
MAXANGLE	Maximum interior angle in degrees (Real > 0.0 see Remark 4 for defaults)
ARATIO	Maximum aspect ratio (Real > 0.0 see Remark 4 for defaults)
EDGERAT	Minimum ratio of the distance between a mid-side node and a corner node and the edge length (Real > 0.0 see Remark 4 for defaults)
BETA	Maximum angle (degrees) between the vector connecting a corner point of a HEXA or PENTA element and the interior node of an adjacent edge and the interior node of the other corner point.(Real > 0.0 see Remark 4 for defaults)
OFFSET	Maximum ratio of the offset distance divided by the length of an element. (Real > 0.0 see Remark 4for defaults)

Remarks

1. Element geometry checking is controlled by the GEOMCHK executive control statement. The values to be used for performing the checks on each element type are specified on the GEOMCHK bulk data entry.
2. Acceptable element names are: BAR, BEAM, HEXA, PENTA, QUAD4, QUAD8, QUADS, SHEAR, TETRA, TRIA3, TRIA6, TRIAS (Currently only the HEXA, PENTA, TETRA, QUAD4, and BEAM are implemented)
3. A separate GEOMCHK entry must be used for each element name specified. Only one GEOMCHK entry may be used for any element name.
4. Default values for the parameters are as follows: (any spot in the table which is blank implies that test is not used for that element type

Table 4-1 Default Values for GEOMCHCK

Name	Skew	Taper	Warp	Min_Angle	Max_Angle	AR	ER	Beta	Offset
Bar									0.15
Beam									0.15
Hexa			45.00	30.00	160.00	50.00	0.10	45.00	

Table 4-1 Default Values for GEOMCHCK

Name	Skew	Taper	Warp	Min_Angle	Max_Angle	AR	ER	Beta	Offset
Penta			45.00	30.00	160.00	50.00	0.10	45.00	
Quad4	40.00	0.60	45.00	30.00	160.00	80.00	0.10		0.00
Quad8	10.00	0.50	45.00	30.00	160.00	50.00	0.10		0.00
Quads	10.00	0.50	45.00	30.00	160.00	50.00	0.10		0.00
Shear	10.00	0.50	45.00			50.00			
Tetra	10.00			30.00	160.00	50.00	0.10		
Tria3	10.00			30.00	160.00	50.00	0.10		
Tria6	10.00			30.00	160.00	50.00	0.10		
Trias	10.00			30.00	160.00	50.00	0.10		

GRAV - *Gravity, or Acceleration, Load*

Description

Defines acceleration load vector.

Format

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	G	N1	N2	N3			

Example

GRAV	1	3	32.2	0.0	0.0	-1.0			

Field	Description
SID	Set identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0)
G	Acceleration vector scale factor. (Real)
N1, N2, N3	Acceleration vector components. (Real; $N1^2 + N2^2 + N3^3 > 0.0$)

Remarks

1. The SID must be unique among all load entries.
2. The gravity, or acceleration, vector is defined by $\vec{g} = G \cdot (N1, N2, N3)$
3. A CID of zero references the basic coordinate system.
4. Gravity loads may be combined with simple loads (for example, FORCEi, MOMENTi) via a LOAD entry.
5. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
6. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.

GRDSET - *Grid Entry Defaults*

Description

Defines CP, CD and PS defaults for GRID entries.

Format

1	2	3	4	5	6	7	8	9	10
GRDSET		CP				CD	PS		

Example

GRDSET		16				32	3456		

Field	Description
CP	Default coordinate system identification number for grid point location definition. (Integer ≥ 0)
CD	Default coordinate system identification number for grid point displacement and solution set representation. (Integer ≥ 0)
PS	Default permanent single-point grid point constraints. (Integer; any unique combination of the integers 1 through 6, with no embedded blanks.)

Remarks

1. Only one GRDSET entry may appear in the Bulk Data section.
2. Defaults provided on this entry will be applied to any GRID entry having corresponding blank field(s). Defaults may be overridden for a grid point by defining either zero or nonzero field values on the GRID entry. (For example, a grid point can be unconstrained by entering a “0” in the PS field, overriding any nonzero value supplied on the GRDSET entry.)
3. The primary purpose of this entry is to minimize the burden of preparing data for problems with a large amount of repetition (for example, two-dimensional pinned-joint problems).
4. At least one of the CP, CD, or PS fields must be specified.

GRID - *Grid Point*

Description

Defines a grid point and its location, coordinate system conventions, and permanent single-point constraints.

Format

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS		

Example

GRID	2	3	1.0	2.0	3.0		316		

Field	Description
ID	Grid point identification number. (Integer > 0)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank; Default = 0, or CP field on GRDSET entry, if present.)
X1, X2, X3	Location of the grid point in coordinate system CP. (Real)
CD	Identification number of coordinate system in which grid point displacements, degrees of freedom, constraints, and solution vectors are defined. (Integer ≥ 0 or blank; Default = 0, or CD field on GRDSET entry, if present.)
PS	Permanent single-point constraints for the grid point. See Remark 3. (Integer ≥ 0 or blank; Default = blank (no constraints), or PS field on GRDSET entry, if present. Any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. Each grid point identification number must be unique with respect to all other structural and scalar points.
2. The meaning of X1, X2, and X3 depend on the type of coordinate system referenced by CP:

Type	Entry	X1	X2	X3
Rectangular	CORDiR	X	Y	Z
Cylindrical	CORDiC	R	θ (degrees)	Z
Spherical	CORDiS	R	θ (degrees)	ϕ (degrees)

3. A nonzero PS field default provided on a GRDSET entry can either be overridden for the current grid point by setting PS (field 8) to a nonzero value, or removed altogether by setting PS to zero.
4. The collection of all CD coordinate systems defined on all GRID entries is called the global coordinate system. All degrees-of-freedom, constraints, and solution vectors are expressed in the global coordinate system.

GTRAN - *Grid Point Transformation*

Description

Used in substructure analysis, this entry defines the output coordinate system transformation to be applied to the displacement set of a selected grid point.

Format

1	2	3	4	5	6	7	8	9	10
GTRAN	SID	NAME	GID	TRANS					

Example

GTRAN	44	GIMBAL	1067	45					

Field	Description
SID	Transformation set identification number. (Integer > 0)
NAME	Basic substructure name. (Character)
GID	Grid point identification number. (Integer > 0)
TRANS	TRANS Bulk Data entry identification number that defines the coordinate system to which displacements will be transformed. See Remark 1. (Integer ≥ 0)

Remarks

1. TRANS options are as follows:
    - If TRANS = 0, the displacements at the grid point will be transformed to the basic coordinate system.
    - If TRANS = SID, the displacements at the grid point will be expressed relative to the combined substructure (i.e., no transformation occurs.)
    - If TRANS = SID of a TRANS entry, displacements at the grid point will be transformed to the coordinate system directions defined by the selected entry.
2. Transformation sets must be selected in Substructure Control (TRAN = SID) to be used by NASTRAN-CORE. Note that TRAN is a subcommand of the substructure COMBINE command.
3. Ensuring that the displacement output for grid point GID is meaningful in the coordinate system defined by the TRANS option is the responsibility of the user.

INCLUDE - *Insert External File*

Description

Inserts an external file into the input file. The INCLUDE entry may appear anywhere within the input data file.

Format

1	2	3	4	5	6	7	8	9	10
INCLUDE	FILNAME								

Example

INCLUDE	'./source_files/myfile.dat'

Field	Description
FILNAME	Physical filename of the external file to be inserted. (Character)

Remarks

1. The INCLUDE statement is a free-format entry with no commas required.
2. The local operating system conventions dictate the syntax used (e.g., file names, delimiters, and path information.) It is recommended that the file path and name string be enclosed by single quotation marks.
3. INCLUDE entries may not be nested; that is, the external file cannot INCLUDE other files.
4. Continuations are not allowed on this entry. The total filename length (including delimiters and quotes) must be 72 characters or less.
5. If a relative path is provided, it will be relative the the directory containing the input file.
6. If no path is specified, the directory will be the one containing the input file.

LOAD - *Static Load Combination (Superposition)*

Description

Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD2, PLOAD4, FORCEAX, MOMAX, PRESAX, SLOAD, RFORCE, and GRAV entries.

Format

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

Example

LOAD	101	-0.5	1.0	3	6.2	4			

Field	Description
SID	Load set identification number. See Remark 2. (Integer > 0)
S	Scale factor. (Real)
Si	Scale factor on Li. (Real)
Li	Static load set identification number specified on one of the above entry types. See Remark 2. (Integer > 0)

Remarks

1. The resultant load vector is given by

$$\{P\} = S\sum_i S_i \cdot \{P_{Li}\}$$

2. SID must be unique with respect to all other LOAD entry SIDs, and with respect to all other static load set identification numbers (Li). Furthermore, LOAD entries may not reference the SIDs of other LOAD entries.
3. This entry must be used if gravity loads (GRAV) are to be combined with any of the other static load types.
4. Load sets must be selected in Case Control (LOAD = SID) to be used by NASTRAN-CORE.

LOADC - *Substructure Static Loading Combination*

Description

Defines the static load for a substructure analysis as a linear combination of load sets defined for each basic substructure.

Format

1	2	3	4	5	6	7	8	9	10
LOADC	SID	S	NAME1	ID1	S1	NAME2	ID2	S2	
			NAME3	ID3	S3	-etc.-			

Example

LOADC	27	1.0	WINGRT	5	0.5	FUSELAG	966	2.5	
			MIDWG	27	1.75				

Field	Description
SID	Load set identification number. (Integer > 0)
S	Scale factor applied to final load vector. (Real)
NAMEi	Basic substructure name. (Character)
IDi	Load set identification number of substructure NAMEi. (Integer > 0)
Si	Scale factor. (Real)

Remarks

1. The load vector is combined by:

$$\{P\} = S\sum_i S_i \cdot \{P_i\}_{IDi}$$

2. The load set identification numbers (IDi) reference the load sets used in Phase 1 to generate the load vectors on the basic substructures.
3. The NAMEi and IDi need not be unique.
4. The LOADC entry is the means of specifying a static loading condition in a Phase 2 substructure analysis. The IDi may actually reference temperature loads or element deformation loads defined in Phase 1.
5. Load sets must be selected in Case Control (LOAD = SID) to be used by NASTRAN-CORE.

MAT1 - *Isotropic Material Property Definition*

Description

Defines the material properties for linear, temperature-independent, isotropic materials.

Format

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

Example

MAT1	17	3.+7	1.9+7		4.28	0.19	5.37+2	0.23	
	20.+4	15.+4	12.+4	2004					

Field	Description
MID	Material identification number. (Integer > 0)
E	Young’s modulus. (Real ≥ 0.0 or blank)
G	Shear modulus. (Real ≥ 0.0 or blank)
NU	Poisson’s ratio. (-1.0 < Real ≤ 0.5 or blank)
RHO	Mass density. (Real)
A	Thermal expansion coefficient. (Real)
TREF	Thermal expansion reference temperature. (Real)
GE	Structural element damping coefficient. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear. (Required for property optimization calculations; otherwise optional if margins of safety are desired.) (Real)
MCSID	Material coordinate system identification number. (Integer ≥ 0 or blank)

Remarks

1. The material identification number must be unique with respect to all other MAT1, MAT2, MAT3 and MAT9 entries.
2. One of E or G must be positive (that is, either E > 0.0 or G > 0.0 or both E and G may be > 0.0).
3. If any one of E, G, or NU is blank, it will be computed to satisfy the identity E = 2(1+NU)G; otherwise, values supplied on this entry will be used.
4. MAT1 materials may be made temperature dependent by use of the MATT1 entry and stress dependent by use of the MATS1 entry.
5. If E and NU or G and NU are both blank they will be given the value 0.0.
6. Weight density may be used in field 6 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
7. Solid elements must not have NU equal to 0.5.
8. Entries for A (thermal expansion coefficient) and TREF (reference temperature) are assumed to be 0.0 when blank.
9. A nonzero MCSID is only required if stresses or strains/curvatures are to be computed in a material coordinate system.

MAT2 - *Anisotropic Material Property Definition*

Description

Defines the material properties for linear, temperature-independent, anisotropic materials.

Format

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
	A1	A2	A12	TREF	GE	ST	SC	SS	
	MCSID								

Example

MAT2	13	6.2+3			6.2+3		5.1+3	0.056	
	0.15			-500.0	0.002	20.+5			
	1008								

Field	Description
MID	Material identification number. (Integer > 0_
Gij	Terms in the material property matrix. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient vector. (Real)
TREF	Thermal expansion reference temperature. (Real)
GE	Structural element damping coefficient. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear. (Used only to compute margins of safety in certain elements; they have no effect on the computational procedures.)
MCSID	Material coordinate system identification number. (Integer ≥ 0 or blank)

Remarks

1. The material identification number must be unique with respect to all other MAT1, MAT2, MAT3 and MAT9 entries.
2. MAT2 materials may be made temperature dependent by use of the MATT2 entry.
3. The convention for the Gij in fields 3 through 8 is represented by the following matrix relationship.

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix}$$

4. A nonzero MCSID is only required if stresses or strains/curvatures are to be computed in a material coordinate system.

MAT3 - *Orthotropic Material Property Definition*

Description

Defines the material properties for linear, temperature-independent, orthotropic materials.

Format

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	EY	EZ	NUXY	NUYZ	NUZX	RHO	
	GXY	GYZ	GZX	AX	AY	AZ	TREF	GE	

Example

MAT3	23	1.0+7	1.1+7	1.2+7	0.3	0.25	0.27	1.0-5	
	2.5+6	3.0+6	2.5+6	1.0-4	1.0-4	1.1-4	68.5	0.23	

Field	Description
MID	Material identification number. (Integer > 0)
EX, EY, EZ	Young’s moduli in the x, y, and z directions. (Real ≥ 0.0)
NUXY, NUYZ, NUZX	Poisson’s Ratios (coupled strain ratios in the xy, yz, and zx directions respectively.) (Real)
RHO	Mass density. (Real)
GXY, GYZ, GZX	Shear moduli for xy, yz, and zx. (Real ≥ 0.0)
AX, AY, AZ	Thermal expansion coefficients. (Real)
TREF	Thermal expansion reference temperature. (Real)
GE	Structural element damping coefficient. (Real)

Remarks

1. The material identification number must be unique with respect to all other MAT1, MAT2, MAT3 and MAT9 entries.
2. MAT3 materials may be made temperature-dependent by use of the MATT3 entry.
3. All nine of the fields EX, EY, EZ, NUXY, NUYZ, NUZX, GXY, GYZ, and GZX must be present.
4. A nonfatal warning message will occur if any of NUXY or NUYZ has an absolute value greater than 1.0.
5. MAT3 materials may only be referenced by CTRIARG, CTRAPRG, CTRIAAX, CTRAPAX, and CTORDRG entries.
6. The mass density, RHO, will be used to automatically compute mass for the CTRIARG, CTRAPRG, CTRIAAX, CTRAPAX, and CTORDRG elements.

MAT8 - *Orthotropic Plate Material Property Definition*

Description

Defines the material property for an orthotropic material for plate elements.

Format

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	XT	XC	YT	YC	S	
	GE	F12							

Example

MAT8	299	32.+6	4.2+5	0.33	2.9+6			0.042	
	14.-6	2.3-6	175.0						
	2.5-4								

Field	Description
MID	Material identification number. (Integer > 0)
E1, E2	Moduli of elasticity in the material x and y directions. (Real $\neq$ 0.0)
NU12	Poisson’s Ratio.(See Remark 5). (Real)
G12	Linear in-plane shear modulus. (Real > 0.0)
G1Z	Transverse shear modulus for shear in X-Z plane. (Real)
G2Z	Transverse shear modulus for shear in Y-Z plane. (Real)
RHO	Mass density. (Real)
A1, A2	Thermal expansion coefficients in the material x and y directions. (Real > 0.0)
TREF	Thermal expansion reference temperature. (Real)
XT, XC	Allowable stresses/strains in tension and compression, respectively, in the material x direction. Required if failure index calculation is desired. See Remark 3. (Real > 0.0; Default value for XC is XT)
YT, YC	Allowable stresses/strains in tension and compression, respectively, in the material y direction. Required if failure index calculation is desired. See Remark 3. (Real > 0.0; Default value for YC is YT)
S	Allowable stress/strain for in-plane shear. See Remark 3. (Real > 0.0)
GE	Structural damping coefficient. (Real)
F12	Tsai-Wu interaction term. See Remark 4. (Real)

Remarks

1. Material coordinate system orientations are defined by the plate element connectivities defined on the CQUAD4 and CTRIA3 entries.
2. The stress-strain relationship defined by this data is:

$$G_{1-z} = \frac{E_{11}}{2(1+\nu_{12})} \quad \text{and} \quad G_{2-z} = \frac{E_{22}}{2(1+\nu_{21})}$$

3. Fields XT, XC, YT, YC, and S are used only for composite materials when failure calculations are requested with PCOMP, PCOMP1, or PCOMP2 Bulk Data entries. Allowables represent stresses except when the maximum strain failure theory is used.
4. The F12 field is used only for composite materials when the Tsai-Wu failure theory is used and failure calculations are requested.
5. NU12 is Poisson's Ratio ($\varepsilon_1/\varepsilon_2$ for uniaxial loading in 1-direction). Note that NU21 = $\varepsilon_1/\varepsilon_2$, uniaxial loading in 2-direction, is related to NU12, E1, and E2 by the relationship, (NU12) (E2) = (NU21) (E1).

MAT9 - Anisotropic Material Property Definition for Solid Elements

Description

Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements. (See PSOLID)

Format

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66	RHO	A1	
	A2	A3	A4	A5	A65	TREF	GE		

Example

MAT9	31	0.23+7	-0.21+7	0.32+6	0.16+7	0.11+7	0.53+6	0.74+7	
	-0.21+7	-0.55+7	-0.37+7	-0.18+7	0.23+7	0.16+7	0.11+7	0.53+6	
	0.66+7	0.28+7	0.14+7	0.43+7	0.92+6	0.30+7	7.32-4		

Field	Description
MID	Material property identification number. (Integer > 0)
Gij	Symmetric portion of 6x6 material matrix. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient vector. (Real)
TREF	Thermal expansion reference temperature. (Real)
GE	Structural damping coefficient. (Real)

Remarks

1. The material property identification number must be unique with respect to all other material property entries.
2. MAT9 materials may be made temperature-dependent by use of the MATT9 entry.
3. The mass density, RHO, is used to compute the structural mass matrix for solid elements.
4. The ordering of the rows and columns of the matrix is critical and must conform to NASTRAN-CORE’s ordering of the stress and strain vectors. The subscripts 1 through 3 refer to the x, y and z axes of the material coordinate system defined by the MCSID field of the PSOLID that references the MID of a MAT9 entry. The material coefficients define a stress strain realtionship in expanded form referred to material coordinates, as follows:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} & G_{14} & G_{15} & G_{16} \\ G_{21} & G_{22} & G_{23} & G_{24} & G_{25} & G_{26} \\ G_{31} & G_{32} & G_{33} & G_{34} & G_{35} & G_{36} \\ G_{41} & G_{42} & G_{43} & G_{44} & G_{45} & G_{46} \\ G_{51} & G_{52} & G_{53} & G_{54} & G_{55} & G_{56} \\ G_{61} & G_{62} & G_{63} & G_{64} & G_{65} & G_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{Bmatrix} (\mathrm{T} - \mathrm{T_{REF}})$$

where T is defined by TEMP entries and T_{REF} is defined on the MAT9 entry.

5. If PARAM W4 is not specified, GE is ignored in tranient analysis.

MATPZ1 - *Piezoelectric Material Property Definition*

Description

Defines the material properties for linear, temperature-independent piezoelectric materials.

Format

1	2	3	4	5	6	7	8	9	10
MATPZ1	MID	S^E_{11}	S^E_{33}	S^E_{44}	S^E_{12}	S^E_{13}	d_{31}	d_{33}	
	d_{15}	S_{ξ_{11}/ξ_0}	S_{ξ_{33}/ξ_0}	RHO	A	TREF	GE		

Example

MATPZ1	1	12.3	15.5	39.0	-4.05	-5.31	-123.0	289.0	
	486.0	730.0	635.0	7500.0					

Field	Description
MID	Material identification number. (Integer > 0)
S^E_{11} thru d_{15}	Piezoelectric constants multiplied by 10^{12}. (Real)
S_{ξ_{11}/ξ_0}, S_{ξ_{33}/ξ_0}	Piezoelectric constants, where ξ_0 is taken to be 8.854×10^{-12} farad/meter. (Real)
RHO	Mass density. (Real)
A	Thermal expansion coefficient. (Real)
TREF	Thermal expansion reference temperature. (Real)
GE	Structural element damping coefficient. (Real)

Remarks

1. MID must be unique with respect to all other material entries.
2. MATPZ1 materials may be made temperature-dependent by use of the MTTPZ1 entry.
3. MATPZ1 may be referenced only by PTRAPAX and PTRIAAX entries.
4. Matrix $[S^E]$ must be nonsingular.

MATPZ2 - *Piezoelectric Material Property Definition*

Description

Defines the material properties for linear, temperature-independent, piezoelectric materials.

Format

1	2	3	4	5	6	7	8	9	10
MATPZ2	MID	C_{11}^E	C_{12}^E	C_{13}^E	C_{14}^E	C_{15}^E	C_{16}^E	C_{22}^E	
	C_{23}^E	C_{24}^E	C_{25}^E	C_{26}^E	C_{33}^E	C_{34}^E	C_{35}^E	C_{36}^E	
	C_{44}^E	C_{45}^E	C_{46}^E	C_{55}^E	C_{56}^E	C_{66}^E	E11	E12	
	E13	E14	E15	E16	E21	E22	E23	E24	
	E25	E26	E31	E32	E33	E34	E35	E36	
	ε_{11}^s	ε_{12}^s	ε_{13}^s	ε_{22}^s	ε_{23}^s	ε_{33}^s	RHO	AX	
	AY	AZ	TREF	GE					

Example

MATPZ2	23	1.	2.	3.	4.	5.	6.	1.	
	2.	3.	4.	5.	1.	2.	3.	4.	
	1.	2.	3.	1.	2.	1.	1.	2.	
	3.	4.	5.	6.	1.	2.	3.	4.	
	5.	6.	1.	2.	3.	4.	5.	6.	
	1.	2.	3.	4.	5.	6.	0.15	6.-7	
	6.-7	6.-7	70.	0.2					

Field	Description
MID	Material identification number. (Integer > 0)
C_{11}^E - ε_{33}^s	Piezoelectric constants. (Real)
RHO	Mass density. (Real)
AX, AY, AZ	Thermal expansion coefficients. (Real)
TREF	Thermal expansion reference temperature. (Real)
GE	Structural element damping coefficient. (Real)

Remarks

1. MID must be unique with respect to all other material entries.
2. MATPZ2 materials may be made temperature-dependent by use of the MTTPZ2 entry.
3. MATPZ2 may be referenced only by PTRAPAX and PTRIAAX entries.
4. See cautionary note, **Section 1.17.3.2**.

MATS1 - *Material Stress Dependence*

Description

Specifies table reference for stress-dependent Young's modulus on a MAT1 entry.

Format

1	2	3	4	5	6	7	8	9	10
MATS1	MID	T(E)							

Example

MATS1	17	28							

Field	Description
MID	Identification number of a MAT1 material entry. (Integer > 0)
T(E)	Reference to table identification number. (Integer ≥ 0)

Remarks

1. A zero or blank field implies no table dependence of the referenced quantity, E, on the basic MAT1 entry. For this case, the MATS1 entry is not required.
2. TABLES1 type tables must be used.

MATT1 - *Isotropic Material Temperature Dependence*

Description

Specifies table references for isotropic material properties on a MAT1 entry that are temperature-dependent.

Format

1	2	3	4	5	6	7	8	9	10
MATT1	MID	T(E)	T(G)	T(NU)	T(RHO)	T(A)		T(GE)	
	T(ST)	T(SC)	T(SS)						

Example

MATT1	17	32				15			
	62								

Field	Description
MID	Identification number of a MAT1 material entry. (Integer > 0)
T(E)	TABLEMi reference for Young's modulus. (Integer ≥ 0 or blank)
T(G)	TABLEMi reference for Shear modulus. (Integer ≥ 0 or blank)
T(NU)	TABLEMi reference for Poisson's ratio. (Integer ≥ 0 or blank)
T(RHO)	TABLEMi reference for mas density. (Integer ≥ 0 or blank)
T(A)	TABLEMi reference for thermal expansion coefficient. (Integer ≥ 0 or blank)
T(GE)	TABLEMi reference for structural damping coefficient. (Integer ≥ 0 or blank)
T(ST)	TABLEMi reference for tensile stress limit. (Integer ≥ 0 or blank)
T(SC)	TABLEMi reference for compressive stress limit. (Integer ≥ 0 or blank)
T(SS)	TABLEMi reference for shear stress limit. (Integer ≥ 0 or blank)

Remarks

1. A blank or zero field implies no table dependence of the referenced quantity on the basic MAT1 entry, and the quantity remains constant.
2. TABLEM1, TABLEM2, TABLEM3, or TABLEM4 type tables may be used to describe the temperature dependency.
3. Material properties given on a basic MATi entry are initial values. If two or more quantities are to retain a fixed relationship, then two or more (as required) tables must be input to define the relationship.

MATT2 - *Anisotropic Material Temperature Dependence*

Description

Specifies table references for anisotropic material properties on a MAT2 entry that are temperature-dependent.

Format

1	2	3	4	5	6	7	8	9	10
MATT2	MID	T(G11)	T(G12)	T(G13)	T(G22)	T(G23)	T(G33)	T(RHO)	
	T(A1)	T(A2)	T(A12)		T(GE)	T(ST)	T(SC)	T(SS)	

Example

MATT2	17	32				15			
	62								

Field	Description
MID	Identification number of a MAT2 material entry. (Integer > 0)
T(Gij)	TABLEMi references for terms in the material property matrix. (Integer ≥ 0 or blank)
T(RHO)	TABLEMi reference for mass density. (Integer ≥ 0 or blank)
TAi)	TABLEMi references for terms in the thermal expansion coefficient vector. (Integer ≥ 0 or blank)
T(GE)	TABLEMi reference for structural element damping coefficient. (Integer ≥ 0 or blank)
T(ST)	TABLEMi reference for stress limit for tension. (Integer ≥ 0 or blank)
T(SC)	TABLEMi reference for stress limit for compression. (Integer ≥ 0 or blank)
T(SS)	TABLEMi reference for stress limit for shear. (Integer ≥ 0 or blank)

Remarks

1. A blank or zero field implies no table dependence of the referenced quantity on the basic MAT2 entry, and the quantity remains constant.
2. TABLEM1, TABLEM2, TABLEM3, or TABLEM4 type tables may be used to describe the temperature dependency.
3. Material properties given on a basic MATi entry are initial values. If two or more quantities are to retain a fixed relationship, then two or more (as required) tables must be input to define the relationship.

MATT3 - *Orthotropic Material Temperature Dependence*

Description

Specifies table references for orthotropic material properties on a MAT3 entry that are temperature-dependent.

Format

1	2	3	4	5	6	7	8	9	10
MATT3	MID	T(EX)	T(EY)	T(EZ)	T(NUXY)	T(NUYZ)	T(NUZX)	T(RHO)	
	T(GXY)	T(GYZ)	T(GZX)	T(AX)	T(AY)	T(AZ)		T(GE)	

Example

MATT3	23	32	48			54			
	74								

Field	Description
MID	Identification number of a MAT3 material entry. (Integer > 0)
T(Ei)	TABLEMi references for Young’s moduli in the x, y, and z directions. (Integer ≥ 0 or blank)
T(NUij)	TABLEMi references for Poisson’s ratios in the xy, yz, and zx directions. (Integer ≥ 0 or blank)
T(RHO)	TABLEMi reference for mass density. (Integer ≥ 0 or blank)
T(Gij)	TABLEMi references for shear moduli in the xy, yz, and zx directions. (Integer ≥ 0 or blank)
T(Ai)	TABLEMi references for thermal expansion coefficients. (Integer ≥ 0 or blank)
T(GE)	TABLEMi reference for structural element damping coefficient. (Integer ≥ 0 or blank)

Remarks

1. A blank or zero field implies no table dependence of the referenced quantity on the basic MAT3 entry, and the quantity remains constant.
2. TABLEM1, TABLEM2, TABLEM3, or TABLEM4 type tables may be used to describe the temperature dependency.
3. Material properties given on a basic MATi entry are initial values. If two or more quantities are to retain a fixed relationship, then two or more (as required) tables must be input to define the relationship.

MATT9 - Temperature Dependence for MAT9 Anisotropic Materail

Description

Provides table references for material properties on a MAT9 entry that are temperature-dependent.

Format

1	2	3	4	5	6	7	8	9	10
MATT9	MID	R11	R12	R13	R14	R15	R16	R22	
	R23	R24	R25	R26	R33	R34	R35	R36	
	R44	R45	R46	R55	R56	R66	RRHO	RA1	
	RA2	RA3	RA4	RA5	RA6		RGE		

Example

MATT9	115	101	102	103	104	105	106	107	
	108	109	110	111	112	113	114	115	
	116	117	118	119	120	121	122	123	
	124	125	126	127	128		130		

Field	Description
MID	Material property identification number of a MAT9 entry. (Integer > 0)
Rij	References to table identification numbers associated with corresponding fields on the MAT9 emtry. (Integer ≥ 0 or blank)
RRHO	References to table identification numbers associated with corresponding RHO field on the MAT9 entry. (Integer ≥ 0 or blank)
RAi	References to table identification numbers associated with corresponding Ai field on the MAT9 entry. (Integer ≥ 0 or blank)
RRHO	References to table identification numbers associated with corresponding GE field on the MAT9 entry. (Integer ≥ 0 or blank)

Remarks

1. Blank or zero entries indicate no table dependence of the referenced MAT9 entry field.
2. TABLEM1, TABLEM2, TABLEM3, and TABLEM4 type tables may be used.
3. Any quantity modifed by MATT9 must have a corresponding value on the referenced MAT9 entry.

MOMAX - *Conical Shell Static Moment*

Description

Defines a static moment loading of a conical shell coordinate.

Format

1	2	3	4	5	6	7	8	9	10
MOMAX	SID	RID	HID	S	MR	MP	MZ		

Example

MOMAX	1	2	3	1.0	0.2	0.3			

Field	Description
SID	Load set identification number. (Integer > 0)
RID	Axisymmetric ring (RINGAX) identification number. (Integer > 0)
HID	Harmonic identification number. See Remark 5. (Integer ≥ 0 or a sequence of harmonics)
S	Scale factor. (Real)
MR, MP, MZ	Moment components in the r, ϕ , z directions. (Real)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Load sets must be selected in Case Control (LOAD = SID) to be used by NASTRAN-CORE.
3. A separate entry is needed for the definition of the moment associated with each harmonic.
4. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
5. A sequence of harmonics can be entered in the HID field using the form, “Sn1Tn2” where n1 is the start of the sequence and n2 is the end of the sequence. For example, “S0T10” indicates loading for harmonics 0 through 10.

MOMENT - *Static Moment*

Description

Defines a static moment at a grid point.

Format

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Example

MOMENT	2	5	6	2.9	0.0	1.0	0.0		

Field	Description
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 or blank; Default = 0)
M	Scale factor. (Real)
N1, N2, N3	Components of vector measured in coordinate system defined by CID.

Remarks

1. The static moment applied at grid point G is given by

$$\vec{m} = M \cdot \vec{N}$$

   where $\vec{N}$ is the vector whose components are defined in fields 6, 7, and 8, and

$$|\vec{m}| = M \cdot |\vec{N}|$$

2. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
3. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
4. A CID of zero or blank references the basic coordinate system.

MOMENT1 - *Static Moment*

Description

Defines a static moment by specification of a magnitude and two grid points which determine the direction.

Format

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

Example

MOMENT1	6	13	-2.93	16	13				

Field	Description
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
M	Value of moment. (Real)
G1, G2	Grid point identification numbers. (Integer >0; G1 and G2 non-coincident)

Remarks

1. The moment defined by this entry is applied at grid point G, and is given by:

$$\vec{m} = M \cdot \vec{N}$$

   where $\vec{N}$ is a unit vector in the direction of G1 to G2, or

$$\vec{N} = \frac{\overrightarrow{G2} - \overrightarrow{G1}}{\left|\overrightarrow{G2} - \overrightarrow{G1}\right|}$$

2. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
3. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.

MOMENT2 - *Static Moment*

Description

Defines a static moment by specification of a magnitude and four grid points which determine the direction.

Format

1	2	3	4	5	6	7	8	9	10
MOMENT2	SID	G	M	G1	G2	G3	G4		

Example

MOMENT2	6	13	-2.93	16	13	17	13		

Field	Description
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
M	Value of moment. (Real)
Gi	Grid point identification numbers. (Integer > 0; G1 and G2 cannot be coincident, nor can G3 and G4.)

Remarks

1. The moment vector defined by this entry is applied at grid point G, and is given by:

$$\vec{m} = M \cdot \vec{N}$$

   where $\vec{N}$ is a unit vector in the direction of the cross product of the vectors G1 to G2, and G3 to G4. The cross product must not be zero.

2. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
3. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.

MPC - *Multipoint Constraint*

Description

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

Format

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc.-				

Example

MPC	3	28	3	6.2	2		4.29		
		1	4	-2.91					

Field	Description
SID	Load set identification number. (Integer > 0)
Gi	Identification number of grid or scalar point. (Integer > 0)
Ci	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
Ai	Coefficient. (Real; Default = 0.0, though A1 must be nonzero)

Remarks

1. The first coordinate in the sequence is assumed to be the dependent degree of freedom and must be unique for all equations of the set.
2. Forces of multipoint constraint are not recovered.
3. Multipoint constraint sets must be selected in Case Control (MPC = SID) to be used by NASTRAN-CORE.
4. The dependent degree of freedom listed on this entry is a member of the *m*-set and may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.

MPCADD - *Multipoint Constraint Set Definition*

Description

Defines a new multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.

Format

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

Example

MPCADD	100	2	3	1	6	4			

Field	Description
SID	Set identification number. (Integer > 0)
Sj	Set identification numbers of multipoint constraint sets defined via MPC entries. (Integer > 0; SID ≠ Sj)

Remarks

1. The Sj must be unique.
2. Multipoint constraint sets must be selected in Case Control (MPC = SID) to be used by NASTRAN-CORE.
3. Sj may not be the identification number of a multipoint constraint set defined by another MPCADD entry.
4. Set identification numbers of 101 or 102 cannot be used in axisymmetric problems.

MPCAX - *Axisymmetric Multipoint Constraint*

Description

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

for a model containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
MPCAX	SID				RID1	HID1	C1	A1	
	RID2	HID2	C2	A2	RID3	HID3	C3	A3	
	-etc.-								

Example

MPCAX	32				17	6	1	1.0	
	23	4	2	-6.8					

Field	Description
SID	Set identification number. (Integer > 0, ≠ 101 or 102)
RIDi	Ring identification number. (Integer > 0)
HIDi	Harmonic identification number. (Integer ≥ 0)
Ci	Component number. (Integer; any one of the integers 1 through 6.)
Ai	Coefficient. (Real; Default = 0.0, though A1 must be nonzero)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. The first coordinate in the sequence is assumed to be the dependent coordinate and must be unique for all equations of the set.
3. Multipoint constraint sets must be selected in Case Control (MPC = SID) to be used by NASTRAN-CORE.
4. The dependent degree of freedom listed on this entry is a member of the *m*-set and may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
5. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
6. For a discussion of the axisymmetric solid problem, see Section 5.11 of the Theoretical Manual.

MPCS - *Substructure Multipoint Constraints*

Description

Defines multipoint constraints within or between substructures.

Format

1	2	3	4	5	6	7	8	9	10
MPCS	SID	NAME1	G1	C1	A1				
		NAME2	G21	C21	A21	G22	C22	A32	
		NAME3	G31	C31	A31	G32	C32	A32	
		-etc.-							

Example

MPCS	171	WINGRT	966	1	1.0				
		FUSELAG	1036	1	0.31	1036	6	32.7	
		CABIN	392	.076					

Field	Description
SID	Load set identification number. (Integer > 0)
NAMEi	Basic substructure name. (Character)
G1, Gij	Grid or scalar point identification numbers in basic substructure NAME1, NAMEi. (Integer > 0)
C1, Cij	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
A1, Aij	Coefficient. (Real; Default = 0.0, though A1 must be nonzero)

Remarks

1. The first degree of freedom in the sequence (defined by NAME1, G1, C1) is the dependent degree of freedom and it must be unique for all equations of the set.
2. MPCS constraints may be imposed only at the SOLVE step of substructuring in Phase 2. The referenced grid point components must therefore exist in the final solution substructure.
3. This entry defines a multipoint constraint equation of the form:

$$A_1 \cdot u_1 + \sum_i \sum_j A_{ij} \cdot u_{ij} = 0$$

   where u1 is the displacement defined by NAME1, G1, and C1, and uij are the displacements defined by NAMEi, Gij, and Cij.
4. Components may be connected within substructures and/or to separate substructures.
5. The dependent degree of freedom listed on this entry is a member of the *m*-set and may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
6. Multipoint constraint sets must be selected in Case Control (MPC = SID) to be used by NASTRAN-CORE.
7. MPCS entries may be referenced by an MPCADD entry.

MTTPZ1 - *Piezoelectric Material Temperature Dependence*

Description

Specifies table references for piezoelectric material properties on a MATPZ1 entry that are temperature-dependent.

Format

1	2	3	4	5	6	7	8	9	10
MTTPZ1	MID	T(S_{11}^E)	T(S_{33}^E)	T(S_{44}^E)	T(S_{12}^E)	T(S_{13}^E)	T(d_{31})	T(d_{33})	
	T(d_{15})	T(S_{ξ_{11}/ξ_0})	T(S_{ξ_{33}/ξ_0})	T(RHO)	T(A)		T(GE)		

Example

MTTPZ1	793	201	202	203	204	205	206	207	
	208	209	210	211	212		213		

Field	Description
MID	Identification number of a MATPZ1 material entry.
T(S_{11}^E) through T(d_{15})	TABLEMi reference for piezoelectric constants. (Integer ≥ 0 or blank)
T(S_{ξ_{11}/ξ_0}), T(S_{ξ_{33}/ξ_0})	TABLEMi reference for piezoelectric constants. (Integer ≥ 0 or blank)
T(RHO)	TABLEMi reference for mass density. (Integer ≥ 0 or blank)
T(A)	TABLEMi reference for thermal expansion coefficient. (Integer ≥ 0 or blank)
T(GE)	TABLEMi reference for structural element damping coefficient. (Integer ≥ 0 or blank)

Remarks

1. A blank or zero field implies no table dependence of the referenced quantity on the basic MATPZ1 entry, and the quantity remains constant.
2. TABLEM1, TABLEM2, TABLEM3, and TABLEM4 type tables may be used to describe the temperature dependence.
3. Material properties given on the basic MATPZ1 entry are initial values. If two or more quantities are to retain a fixed relationship, then two or more tables must be input to define the relationship.

MTTPZ2 - *Piezoelectric Material Temperature Dependence*

Description

Specifies table references for piezoelectric material properties on a MATPZ2 entry that are temperature-dependent.

Format

1	2	3	4	5	6	7	8	9	10
MTTPZ2	MID	T(C_{11}^{E})	T(C_{12}^{E})	T(C_{13}^{E})	T(C_{14}^{E})	T(C_{15}^{E})	T(C_{16}^{E})	T(C_{22}^{E})	
	T(C_{23}^{E})	T(C_{24}^{E})	T(C_{25}^{E})	T(C_{26}^{E})	T(C_{33}^{E})	T(C_{34}^{E})	T(C_{35}^{E})	T(C_{36}^{E})	
	T(C_{44}^{E})	T(C_{45}^{E})	T(C_{46}^{E})	T(C_{55}^{E})	T(C_{56}^{E})	T(C_{66}^{E})	T(E11)	T(E12)	
	T(E13)	T(E14)	T(E15)	T(E16)	T(E21)	T(E22)	T(E23)	T(E24)	
	T(E25)	T(E26)	T(E31)	T(E32)	T(E33)	T(E34)	T(E35)	T(E36)	
	T(ε_{11}^{s})	T(ε_{12}^{s})	T(ε_{13}^{s})	T(ε_{22}^{s})	T(ε_{23}^{s})	T(ε_{33}^{s})	T(RHO)	T(AX)	
	T(AY)	T(AZ)		T(GE)					

Field	Description
MID	Identification number of a MATPZ2 material entry. (Integer > 0)
T(C_{11}^{E}) through T(ε_{33}^{s})	TABLEMi reference for piezoelectric constants. (Integer ≥ 0 or blank)
T(RHO)	TABLEMi reference for mass density. (Integer ≥ 0 or blank)
T(AX), T(AY), T(AZ)	TABLEMi reference for thermal expansion coefficients. (Integer ≥ 0 or blank)
T(GE)	TABLEMi reference for structural element damping coefficient. (Integer ≥ 0 or blank)

Remarks

1. A blank or zero field implies no table dependence of the referenced quantity on the basic MATPZ2 entry, and the quantity remains constant.
2. TABLEM1, TABLEM2, TABLEM3, and TABLEM4 type tables may be used to describe the temperature dependence.
3. Material properties given on the basic MATPZ2 entry are initial values. If two or more quantities are to retain a fixed relationship, then two or more tables must be input to define the relationship.

NOLIN1 - *Nonlinear Transient Response Dynamic Load*

Description

Defines nonlinear transient forcing functions of the form

$$P_i(t) = ST(x_j(t)) ,$$

where x_j is either a displacement (u_j) or a velocity $(\dot{u}_j)$.

Format

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	GI	CI	S	GJ	CJ	TID		

Example

NOLIN1	21	3	4	2.1	3	1	6		

Field	Description
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number if GJ is a grid point. See Remark 4. (Integer ≥ 0 or blank)
T	Identification number of a TABLEDi entry. (Integer > 0)

Remarks

1. Nonlinear loads must be selected in Case Control (NONLINEAR = SID) to be used by NASTRAN-CORE.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All coordinates referenced on NOLIN1 entries must be members of the solution set. This means the u_e set for modal formulation and the $u_d = u_e + u_a$ set for direct formulation.
4. The permissible values for the component number CJ are given in the following table:

x_j	Grid point, GJ	Scalar or extra point, GJ
Displacement (u_j)	1 ≤ Integer ≤ 6	0 or blank
Velocity $(\dot{u}_j)$	11 ≤ Integer ≤ 16	10

   Note that velocity components are represented by integers ten greater than the corresponding displacement components.

5. If x_j is a velocity $(\dot{u}_j)$, then it is determined from the relation

$$\dot{u}_{j,t} = \frac{(u_{j,t} - u_{j,t-1})}{\Delta t}$$

where Δt is the time increment and $u_{j,\,t}$ and $u_{j,\,t-1}$ are the displacements at time t and at the previous time step respectively.

NOLIN2 - *Nonlinear Transient Response Dynamic Load*

Description

Defines nonlinear transient forcing functions of the form

$$P_i(t) = Sx_j(t)y_k(t),$$

where x_j and y_k are either displacements (u_j, u_k) or velocities $(\dot{u}_j, \dot{u}_k)$.

Format

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	GI	CI	S	GJ	CJ	GK	CK	

Example

NOLIN2	14	2	1	2.9	2	1	2	11	

Field	Description
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid or scalar or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number if GJ is a grid point. See Remark 4. (Integer ≥ 0 or blank)
GK	Grid, scalar, or extra point identification number.
CK	Component number if GK is a grid point. See Remark 4. (Integer ≥ 0 or blank)

Remarks

1. Nonlinear loads must be selected in Case Control (NONLINEAR = SID) to be used by NASTRAN-CORE.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All coordinates referenced on NOLIN2 entries must be members of the solution set. This means the u_e set for modal formulation and the $u_d = u_e + u_a$ set for direct formulation.
4. The permissible values for the component number CJ or CK are given in the following table:

x_j **or** y_k	**Grid point, GJ or GK**	**Scalar or extra point, GJ or GK**
Displacement (u_j or u_k)	1 ≤ Integer ≤ 6	0 or blank
Velocity ($\dot{u}_j$ or $\dot{u}_k$)	11 ≤ Integer ≤ 16	10

   Note that velocity components are represented by integers ten greater than the corresponding displacement components.

5. If x_j or y_k is a velocity $(j \text{ or } k)$, then it is determined from the relation

$$\dot{u}_{j,t} = \frac{(u_{j,t} - u_{j,t-1})}{\Delta t} \qquad \text{or} \qquad \dot{u}_{k,t} = \frac{(u_{k,t} - u_{k,t-1})}{\Delta t}$$

where Δt is the time increment, $u_{j,t}$ and $u_{k,t}$ are the displacements at the time t and $u_{j,t-1}$ and $u_{k,t-1}$ are the displacements at the previous time step.

6. x_j and y_k need not both represent displacements or velocities. One of them may be a displacement and the other may be a velocity.

NOLIN3 - *Nonlinear Transient Response Dynamic Load*

Description

Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} S(x_j(t))^A, & x_j(t) > 0 \\ 0, & x_j(t) \le 0 \end{cases},$$

where xj is either a displacement (u_j) or a velocity $(\dot{u}_j)$.

Format

1	2	3	4	5	6	7	8	9	10
NOLIN3	SID	GI	CI	S	GJ	CJ	A		

Example

NOLIN3	4	102		-6.1	2	5	-3.5		

Field	Description
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number if GJ is a grid point. See Remark 4. (Integer ≥ 0 or blank)
A	Amplification factor. (Real)

Remarks

1. Nonlinear loads must be selected in Case Control (NONLINEAR = SID) to be used by NASTRAN-CORE.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All coordinates referenced on NOLIN3 entries must be members of the solution set. This means the u_e set for modal formulation and the $u_d = u_e + u_a$ set for direct formulation.
4. The permissible values for the component number CJ are given in the following table:

x_j	Grid point, GJ	Scalar or extra point, GJ
Displacement (u_j)	$1 \le$ Integer ≤ 6	0 or blank
Velocity $(\dot{u}_j)$	$11 \le$ Integer ≤ 16	10

   Note that velocity components are represented by integers ten greater than the corresponding displacement components.

5. If x_j is a velocity $(\dot{u}_j)$, then it is determined from the relation: $\dot{u}_{j,t} = \frac{(u_{j,t} - u_{j,t-1})}{\Delta t}$
   where Δt is the time increment and $u_{j,t}$ and $u_{j,t-1}$ are the displacements at time t and at the previous time step, respectively.

NOLIN4 - *Linear Transient Response Dynamic Load*

Description

Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} -S(-x_j(t))^A, x_j(t) < 0 \\ 0, x_j(t) \geq 0 \end{cases},$$

where x_j is either a displacement (u_j) or a velocity $(\dot{u}_j)$.

Format

1	2	3	4	5	6	7	8	9	10
NOLIN4	SID	GI	CI	S	GJ	CJ	A		

Example

NOLIN4	2	4	6	2.0	101		16.3		

Field	Description
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number.
CJ	Component number if GJ is a grid point. See Remark 4. (Integer ≥ 0 or blank)
A	Amplification factor. (Real)

Remarks

1. Nonlinear loads must be selected in Case Control (NONLINEAR = SID to be used by NASTRAN-CORE.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All coordinates referenced on NOLIN4 entries must be members of the solution set. This means the u_e set for modal formulation and the $u_d = u_e + u_a$ set for direct formulation.
4. The permissible values for the component number CJ are given in the following table:

x_j	Grid point, GJ	Scalar or extra point, GJ
Displacement (u_j)	1 ≤ Integer ≤ 6	0 or blank
Velocity $(\dot{u}_j)$	11 ≤ Integer ≤ 16	10

   Note that velocity components are represented by integers ten greater than the corresponding displacement components.
5. If x_j is a velocity $(\dot{u}_j)$, then it is determined from the relation

$$\dot{u}_{j,t} = \frac{(u_{j,t} - u_{j,t-1})}{\Delta t}$$

where Δt is the time increment and $u_{j,t}$ and $u_{j,t-1}$ are the displacements at time t and at the previous time step, respectively.

NOLIN6 - *Nonlinear Transient Response Dynamic Load*

Description

Defines nonlinear transient forcing functions of the form

$$P_i(t) = ST(x_j(t))\left| x_j(t) \right| x_j(t) \qquad \text{if } CJ \le 6$$
$$P_i(t) = ST(x_j(t))\left| \dot{x}_j(t) \right| \dot{x}_j(t) \qquad \text{if } CJ \ge 10$$

Format

1	2	3	4	5	6	7	8	9	10
NOLIN6	SID	GI	CI	S	GJ	CJ	TID		

Example

NOLIN6	21	3	4	2.1	3	1	6		

Field	Description
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid or scalar or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
S	Scale factor. (Real)
GJ	Grid or scalar or extra point identification number. (Integer > 0)
CJ	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.) See Remark 4.
TID	Identification number of a TABLEDi entry. (Integer > 0)

Remarks

1. Nonlinear loads must be selected in Case Control (NONLINEAR = SID) to be used by NASTRAN-CORE.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All coordinates referenced on NOLIN6 entries must be members of the solution set. This means the u_e set for modal formulation and the $u_d = u_e + u_a$ set for direct formulation.
4. The permissible values for the component number CJ are given in the following table:

x_j or $\dot{x}_j$	Grid point, GJ	Scalar or extra point, GJ
Displacement (x_j)	1 ≤ Integer ≤ 6	0 or blank
Velocity ($\dot{x}_j$)	11 ≤ Integer ≤ 16	10

Note that velocity components are represented by integers ten greater than the corresponding displacement components.

5. Velocity $\dot{x}_j$ is determined from the relation:

$$\dot{x}_{j,t} = \frac{(x_{j,t} - x_{j,t-1})}{\Delta t}$$

where Δt is the time increment and $x_{j,\,t}$ and $x_{j,\,t-1}$ are the displacements at time t and at the previous time step respectively.

6. Since the forcing function $P_i(t)$ is a product of TABLEDi, displacement, velocity and the scale factor S, any zero value of these quantities will make $P_i(t)$ equal to zero. This condition may occur when the initial displacements or velocities are zero, and no other load is applied to the structure.

NOTMSET - *Definition of DOF which may not be placed in the m-set*

Description

Defines degrees of freedom which may not be placed in the dependent set (m-set) during the auto M-set operations..

Format

1	2	3	4	5	6	7	8	9	10
NOTMSET	ID	C	ID	C	ID	C	ID	C	

Example

NOTMSET	16	2	23	3516			1	4	

Field	Description
ID	Grid or scalar point identification numbers.
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. Degrees of freedom selected on this entry are placed into the *notm*-set, and may be referenced on entries that define other mutually-exclusive sets.
2. When NOTMSET or NOTMSET1 entries are present, the degrees of freedom listed on these entries may not be placed into the *m*-set by the MCE1 module when it is performing the automatic M-set definition.

NOTMSET1 - *Definition of DOF which may not be placed in the m-set, Alternate Form*

Description

Defines degrees of freedom which may not be placed in the dependent set (m-set) during the auto M-set operations.. Alternate form of the NOTMSET entry.

Format

1	2	3	4	5	6	7	8	9	10
NOTMSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example

NOTMSET1	345	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example

NOTMSET1	C	ID1	“THRU”	ID2					
NOTMSET1	123456	7	THRU	109					

Field	Description
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0)

Remarks

1. Degrees of freedom selected on this entry are placed into the *notm*-set, and may be referenced on entries that define other mutually-exclusive sets.
2. When NOTMSET or NOTMSET1 entries are present, the degrees of freedom listed on these entries may not be placed into the *m*-set by the MCE1 module when it is performing the automatic M-set definition.
3. If this alternate form is used, all of the grid or scalar points ID1 through ID2 must be defined.

OMIT - *Omitted Coordinates*

Description

Defines degrees of freedom to be omitted from the solution set via matrix partitioning. Used to reduce the number of independent degrees of freedom.

Format

1	2	3	4	5	6	7	8	9	10
OMIT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example

OMIT	16	2	23	3516			1	4	

Field	Description
ID	Grid or scalar point identification numbers. (Integer > 0)
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. Degrees of freedom specified on OMIT entries belong to the omitted degree of freedom set (*o*-set.) The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
2. As many as 24 coordinates may be omitted by a single entry.

OMIT1 - *Omitted Coordinates*

Description

Defines degrees of freedom to be omitted from the solution set via matrix partitioning. Used to reduce the number of independent degrees of freedom.

Format

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	-etc.-						

Example

OMIT1	3	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example

OMIT1	C	G1	“THRU”	G2					
OMIT1	0	17	THRU	109					

Field	Description
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Gi	Grid or scalar point identification numbers. (Integer > 0; G1 < G2 when using the “THRU” option)

Remarks

1. Degrees of freedom specified on OMIT entries belong to the omitted degree of freedom set (*o*-set.) The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
2. If the alternate format is used, all of the grid (or scalar) points G1 through G2 inclusive must be defined.

OMITAX - *Axisymmetric Omitted Coordinates*

Description

Defines coordinates to be omitted from a model containing CCONEAX, CTRAPAX, or CTRIAAX elements through matrix partitioning. Used to reduce the number of independent degrees of freedom.

Format

1	2	3	4	5	6	7	8	9	10
OMITAX	RID	HID	C	RID	HID	C			

Example

OMITAX	2	6	3	4	7	1			

Field	Description
RID	Axisymmetric ring (RINGAX) identification number. (Integer > 0)
HID	Harmonic identification number. (Integer ≥ 0)
C	Component number(s). (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Up to 12 coordinates may be omitted via this entry.
3. Coordinates appearing on OMITAX entries may not appear on MPCAX, SUPAX, or SPCAX entries.
4. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
5. For a discussion of the axisymmetric solid problem, see Section 5.11 of the Theoretical Manual.

PARAM - *Parameter Value Definition*

Description

Specifies values for parameters appearing in the DMAP solution sequences.

Format

1	2	3	4	5	6	7	8	9	10
PARAM	N	V1	V2						

Example

PARAM	IRES	1							

Field	Description
N	Parameter name. (Character)
V1, V2	Parameter value based on parameter type (declared in the DMAP) as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single-precision	Real	Blank
Character	Character	Blank
Real, double-precision	Double-precision	Blank
Complex, single-precision	Real	Real
Complex, double-precision	Double-precision	Double-precision

Remarks

1. **Section 5** describes parameters used in the DMAP solution sequences.
2. See Section 4, Parameters, for a complete list of parameters, their types, usage, and defaults.

PBAR - *Simple Beam Properties*

Description

Defines the properties for simple beam (bar) elements, defined via the CBAR entry.

Format

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

Example

PBAR	39	6	2.9		4.97				
			2.0	4.0					

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
A	Area of bar cross-section. (Real)
I1, I2, I12	Area moments of inertia. (Real; I1 ≥ 0.0, I2 ≥ 0.0, $I1 \cdot I2 \geq (I12)^2$
J	Torsional constant. (Real)
NSM	Nonstructural mass per unit length. (Real)
K1, K2	Area factor for shear. See Remark 4. (Real or blank)
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real)

Remarks

1. PBAR entries must have unique identification numbers with respect to all other property entries.
2. For structural problems, PBAR entries may only reference MAT1 material entries.
3. See **Section 1.3.2** for a discussion of bar element geometry.

4. The quantities K1 and K2 are expressed as the relative amounts (0.0 to 1.0) of the total cross-sectional area contributing to the transverse shear stiffness (KAG) in the direction of the two principal axes. These quantities are ignored if I12 is non-zero. Defaults for K1 and K2 are: K1 = (12*E*I1)/(L*L*L); K2 = (12*EII2)/(L*L*L).

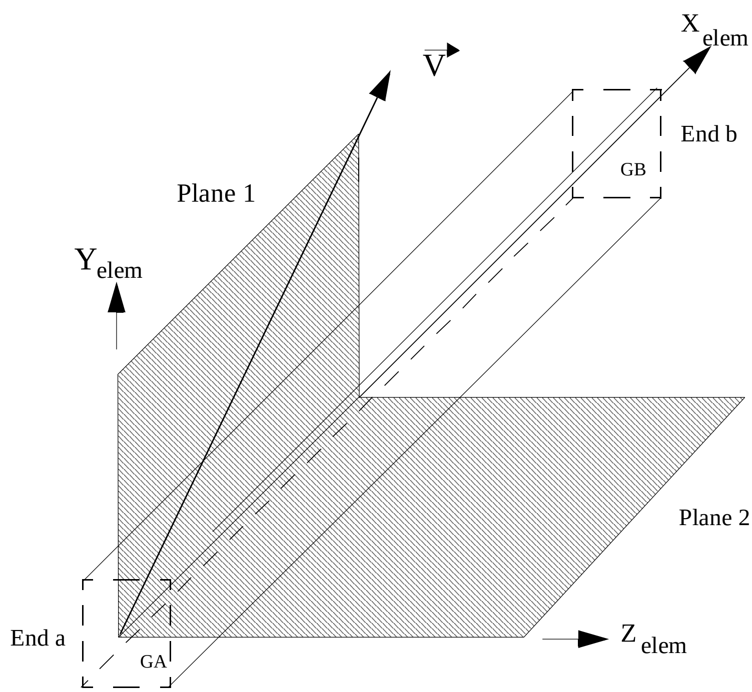


Figure 1. PBAR Element Coordinate System

PBARL - *Simple Beam Properties defined using Cross-Section Dimensions*

Description

Defines the properties for simple beam (bar) elements, defined via the CBAR entry.

Format

1	2	3	4	5	6	7	8	9	10
PBARL	PID	MID		TYPE					
	DIM1	DIM2	DIM3	etc	DIMn	NSM			

Example

PBARL	39	6		BAR					
	1.	1.							

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
TYPE	Name of cross-section used (Character: - allowble names are: BAR, BOX, BOX1, CHAN, CHAN1, CHAN2, CROSS, H, HAT, HEXA, I, I1, ROD, T, T1, T2, TUBE, AND Z)
DIMi	Dimensions for the selected cross-section (Real, > 0.0) see remark 6
NSM	Nonstructural mass per unit length. (Real, default = 0.0)

Remarks

1. PBARL entries must have unique identification numbers with respect to all other property entries.
2. For structural problems, PBARL entries may only reference MAT1 material entries.
3. See **Section 1.3.2** for a discussion of bar element geometry.
4. Note that BAR elements do not account for offsets between the neutral axis and the shear center. Unsymmetric cross-sections, may therefore give inorrect answers, depending on the loading. If you are using unsymmetric cross sections, the BEAM element accounts for the offset.
5. PBARL entries result in the creation of “PBAR” records in the EPT, containing the calculated values. This is done by the BEAMLIB DMAP module.
6. Section names and their associated dimensions and data recovery locations are shown in **Figure 4-34**

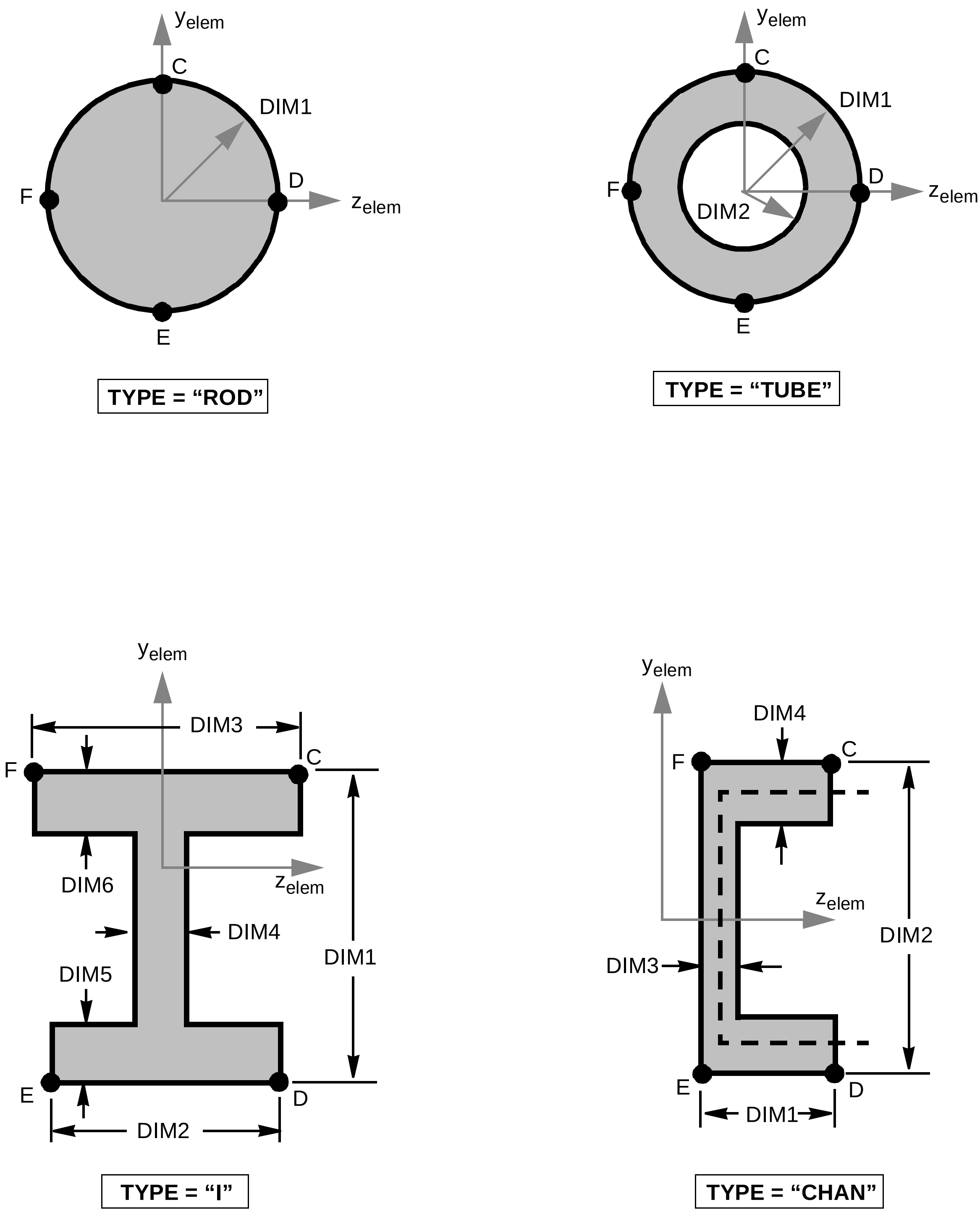
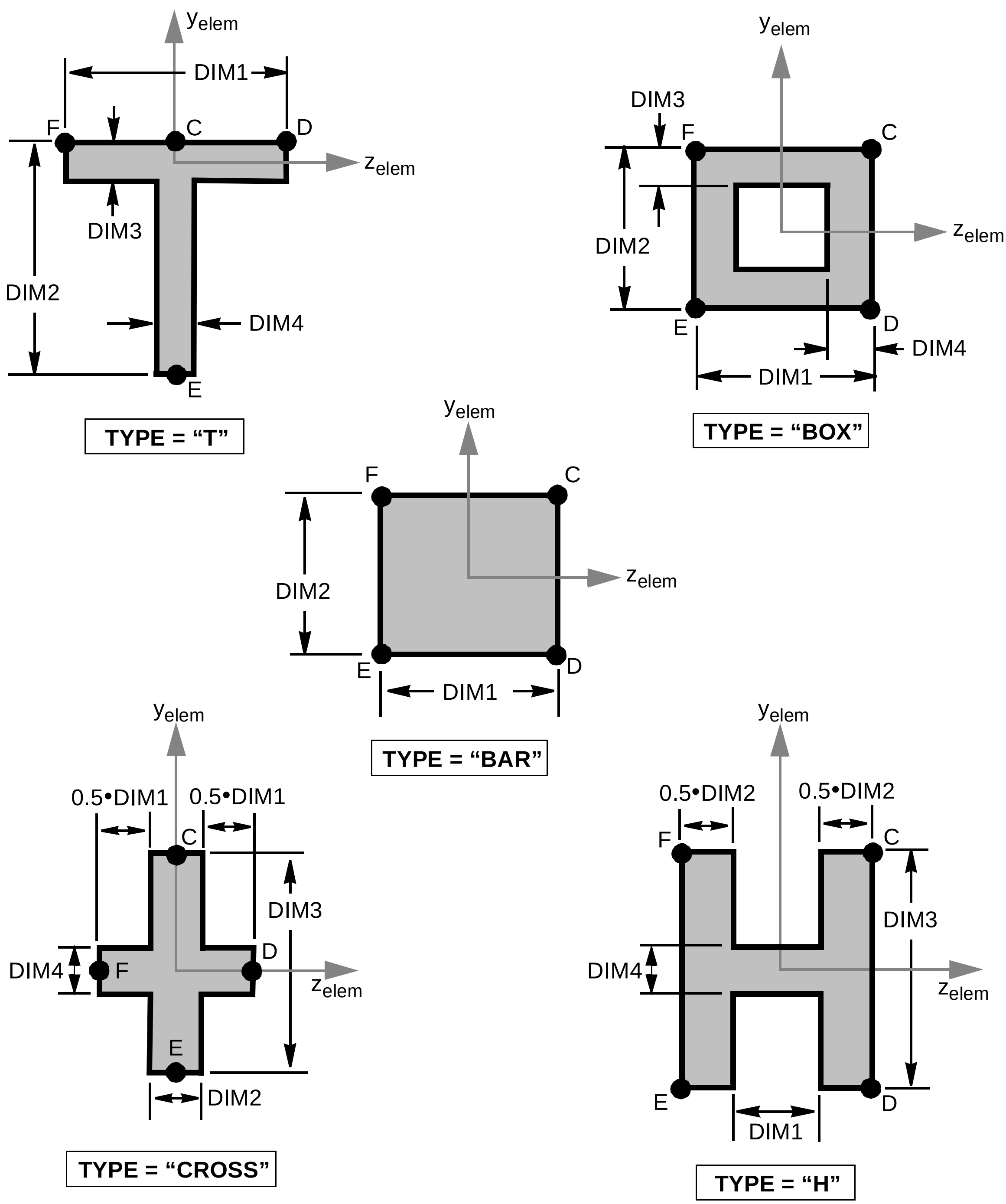
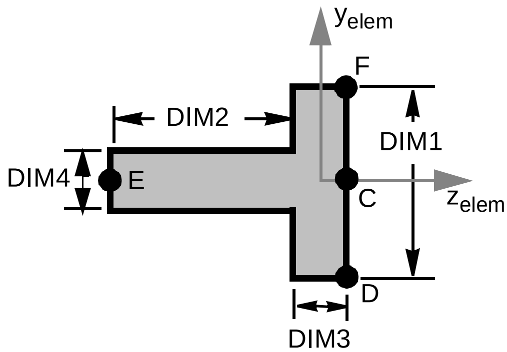


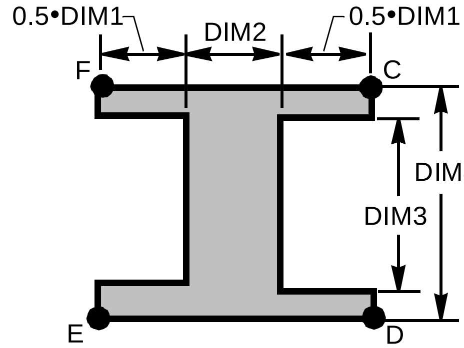
Figure 4-34 Dimensions and Data Recovery Locations for PBARL Cross-Sections



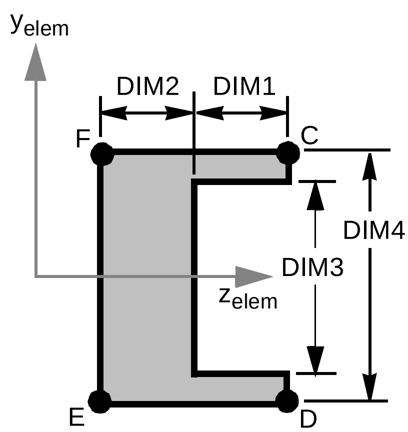
Dimensions and Data Recovery Locations for PBARL Cross-Sections (continued)



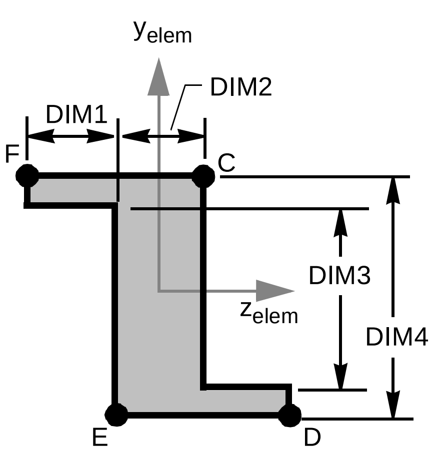
TYPE = "T1"



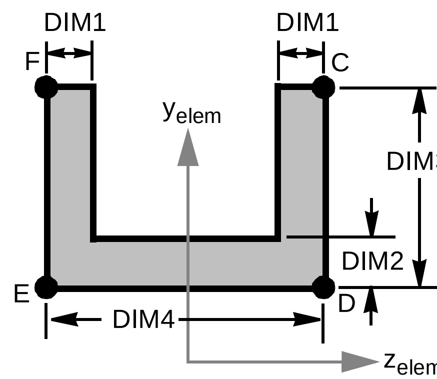
TYPE = "I1"



TYPE = "CHAN1"

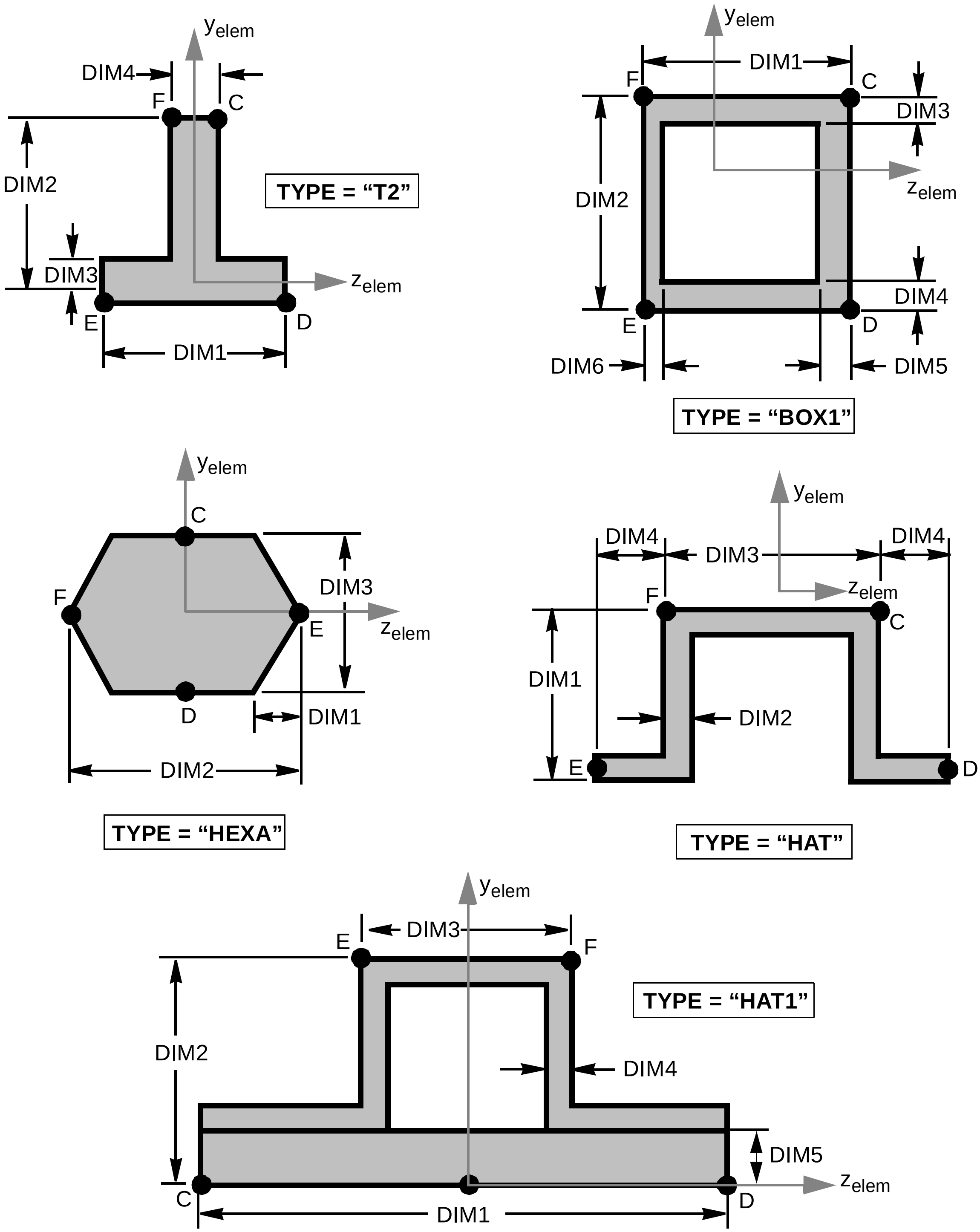


TYPE = "Z"



TYPE = "CHAN2"

Dimensions and Data Recovery Locations for PBARL Cross-Sections (continued)



Dimensions and Data Recovery Locations for PBARL Cross-Sections (continued)

PBEAM - *Beam Properties*

Description

Defines the properties for beam elements, defined via the CBEAM entry.

Format

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A	$I1_A$	$I2_A$	$I12_A$	J_A	NSM_A	
	$C1_A$	$C2_A$	$D1_A$	$D2_A$	$E1_A$	$E2_A$	$F1_A$	$F2_A$	

The next two continuations may be repeated for up to 9 intermediate stations as described in Remark 3, where SO and X/XB must be specified for any sections defined

	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

	K1	K2	S1	S2	NSI_A	NSI_B	CW_A	CW_B	
	M_A1	$M2_A$	M_B1	$M2_B$	$N1_A$	$N2_A$	$N1_B$	$N2_B$	

Example

PBEAM	39	6	2.9		4.97				
			2.0	4.0					

Field	Description	Default
PID	Property identification number. (Integer > 0)	Required
MID	Material identification number (see remark 1). (Integer > 0)	Required
A_A	Area of bar cross-section at end A of the BEAM. (Real ≥ 0.0)	0.0
$I1_A$, $I2_A$, $I12_A$	Area moments of inertia and product of inertia at end A of the BEAM. (Real; I1 ≥ 0.0, I2 ≥ 0.0, $I1 \cdot I2 \geq (I12)^2$	0.0
J_A	Torsional constant at end A of the BEAM. (Real ≥ 0.0)	0.0
NSM_A	Nonstructural mass per unit length at end A of the BEAM. (Real ≥ 0.0)	0.0
Ci_A, Di_A, Ei_A, Fi_A	Stress recovery coefficients at end A of the BEAM. (Real)	0.0
SO	Stress output request option. Must be one of the character strings 'YESA', YES', or 'NO'. There is no default.	Required if continuation is used
X/XB	Distance from end A in the element coordinate system as a fraction of the total length of the BEAM.	Required if continuation is used

Field	Description	Default
I1, I2, I12	Area moments of inertia and product of inertia at the intermediate station. (Real; I1 ≥ 0.0 or blank, I2 ≥ 0.0 or blank, $I1 \cdot I2 \geq (I12)^2$	0.0
J	Torsional constant at the intermediate station. (Real ≥ 0.0 or blank)	0.0
NSM	Nonstructural mass per unit length at the intermediate station. (Real ≥ 0.0 or blank)	0.0
K1, K2	Area factor for shear in planes 1 and 2 respectively. See Remark 6. (Real ≥ 0.0 or blank)	0.0
S1, S2	Ignored.(Real ≥ 0.0 or blank)	0.0
NSI_A, NSI_B	ignored.Real ≥ 0.0 or blank)	0.0
CW_A, CW_B	ignored.Real ≥ 0.0 or blank)	0.0
$M1_A$, $M1_B$, $M2_A$, $M2_B$,	ignored.Real ≥ 0.0 or blank)	0.0
$N1_A$, $N1_B$, $N2_A$, $N2_B$,	ignored.Real ≥ 0.0 or blank)	0.0

Remarks

1. MID on PBEAM entries may only reference MAT1 material entries.
2. See the CBEAM description for a discussion of beam element geometry and output.
3. The second and third continuations (Starting with SO and ending with F2) must appear in sets and as many as 9 internediate stations may be defined. For each intermediate station, SO and X/XB must be specified. If intermediate stations are defined, the final one must have X/XB = 1.0.
4. If no stress output is desired at end A and a conitnuation entry starting with SO is specified, then the continuation entry containing $C1_A$ through $F2_A$ may be omitted.
5. If, for any station, SO is YESA or NO then the continuation line for that section containing C1 through F2 must be omitted. If SO is YES for any station, the continuation containing C1 through F2 is required for that cross-section.
6. The quantities K1 and K2 are expressed as the relative amounts (0.0 to 1.0) of the total cross-sectional area contributing to the transverse shear stiffness (KAG) in the direction of the two principal axes. These quantities are ignored if I12 is non-zero. Defaults for K1 and K2 are 1. Note that the default for these factors on the PBAR are zero. A value of 0.0 for either of these means that the associated transverse shear flexibility is omitted.
7. If an intermediate station does not have values entered for any of the fields (A, I1, I2, I12, J, NSM, C1, C2, D1, D2, E1, E2, F1, F2), the program will use linear interpolation between the values for the nearest stations containing non-zero values for the associated data.
8. If on the station for X/XB=1.0, no properties are specified, the station will use the values from end A.
9. **Figure 4-35** Shows the BEAM element coordinate system - the input properties for the PBEAM correspond to:

Table 4-2 BEAM properties

On PBEAM Entry	Corresponding Property of element
I1	I_{ZZ} about neutral axis
I22	I_{YY} about neutral axis
$N1_A$	Y_{elem} distance from shear center to neutral axis at end A
$N2_A$	Z_{elem} distance from shear center to neutral axis at end A
$N1_B$	Y_{elem} distance from shear center to neutral axis at end B
$N2_B$	Z_{elem} distance from shear center to neutral axis at end B

Table 4-2 BEAM properties

On PBEAM Entry	Corresponding Property of element
$M1_A$, $M1_B$	Y_{elem} distance from shear center to nonstructural mass center of gravity at ends A and B respectively
$M2_A$, $M2_B$	Z_{elem} distance from shear center to nonstructural mass center of gravity at ends A and B respectively

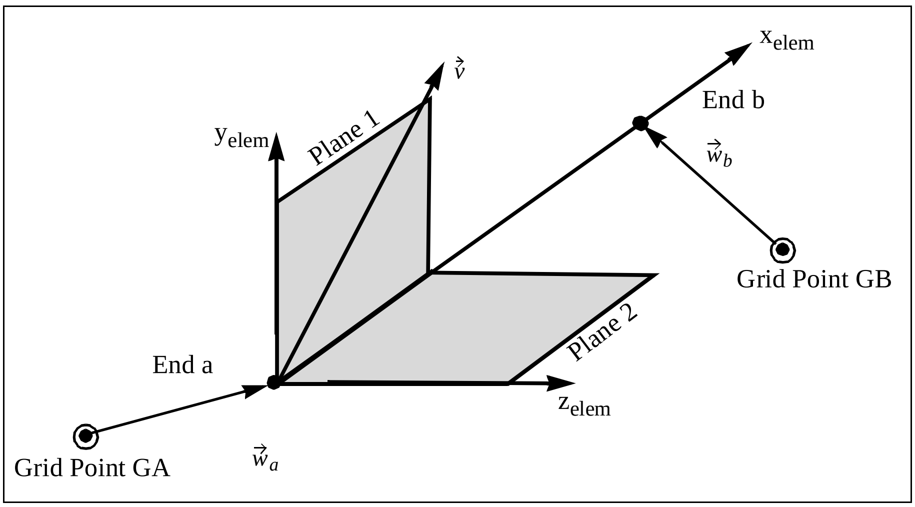


Figure 4-35 BEAM orientation

10.

PBEAML - *Beam Properties defined using Cross-Section Dimensions*

Description

Defines the properties for beam elements, defined via the CBEAM entry.

Format

1	2	3	4	5	6	7	8	9	10
PBEAML	PID	MID		TYPE					
	DIM1(A)	DIM2(A)	DIM3(A)	etc	DIMn(A)	NSM(A)	SO(1)	X(1)/XB	
	DIM1(1)	DIM2(1)	etc	DIMn(1)	NSM(1)	SO(2)	DIM1(2)	DIM2(2)	
	etc	DIMn(2)	etc	DIMn(2)	NSM(2)	etc	SO(j)	DIM1(j)	
	DIM2(j)	etc	DIMn(j)	NSM(j)	SO(B)	DIM1(B)	DIM2(B)	etc	
	DIMn(B)	NSM(B)							

Example

PBEAML	39	6		BAR					
	1.	1.							

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
TYPE	Name of cross-section used (Character: - allowble names are: BAR, BOX, BOX1, CHAN, CHAN1, CHAN2, CROSS, H, HAT, HEXA, I, I1, L, ROD, T, T1, T2, TUBE, AND Z)
DIMi	Dimensions for the selected cross-section (Real, > 0.0)
NSM	Nonstructural mass per unit length. (Real, default = 0.0)

Remarks

1. PBEAML entries must have unique identification numbers with respect to all other property entries.
2. For structural problems, PBEAML entries may only reference MAT1 material entries.
3. See the CBEAM and PBEAM entries for a discussion of beam element geometry.
4. PEAML entries result in the creation of “PBEAM” records in the EPT, containing the calculated values. This is done by the BEAMLIB DMAP module.
5. Dimensions and data recovery locations are shown in **Figure 4-36**

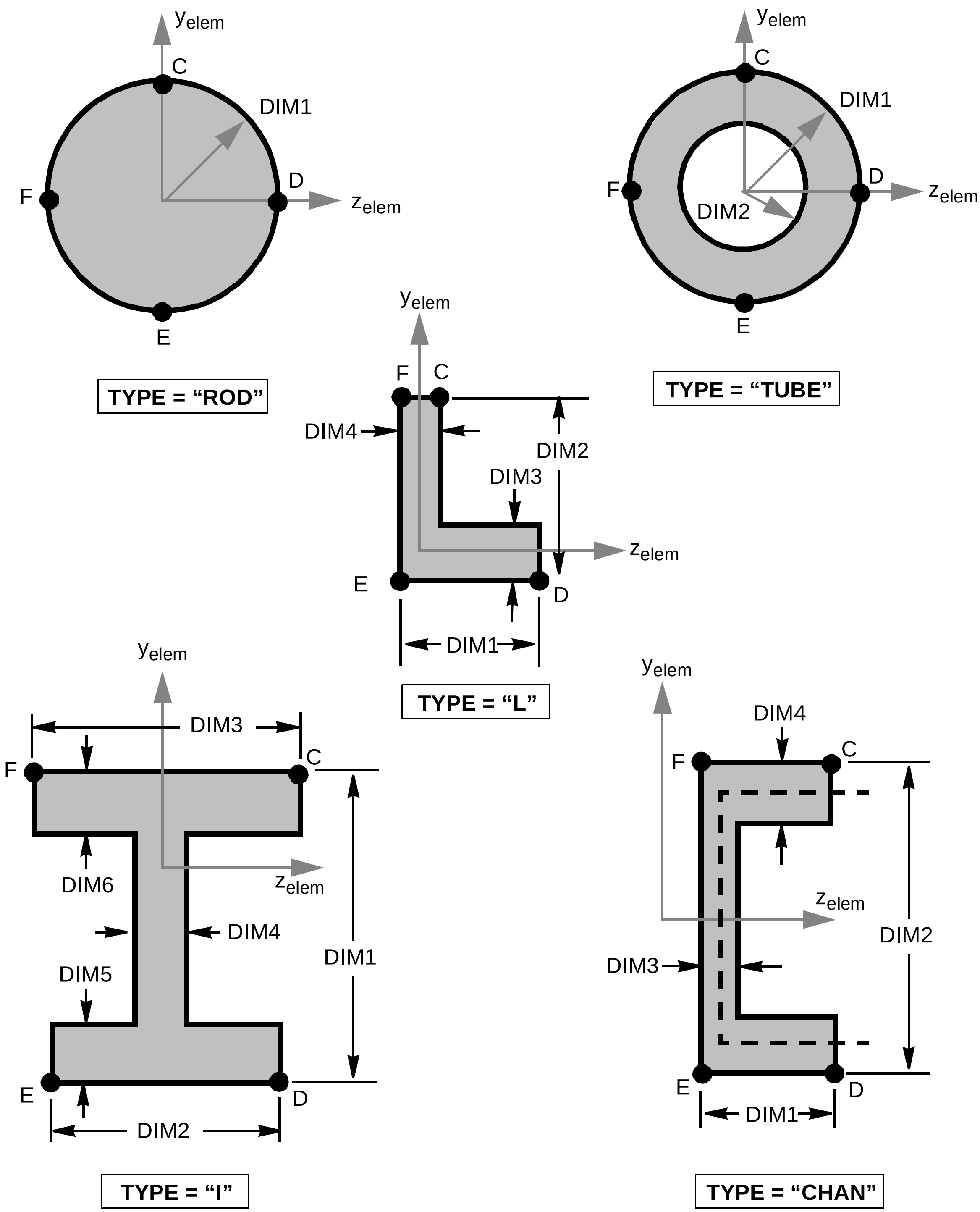
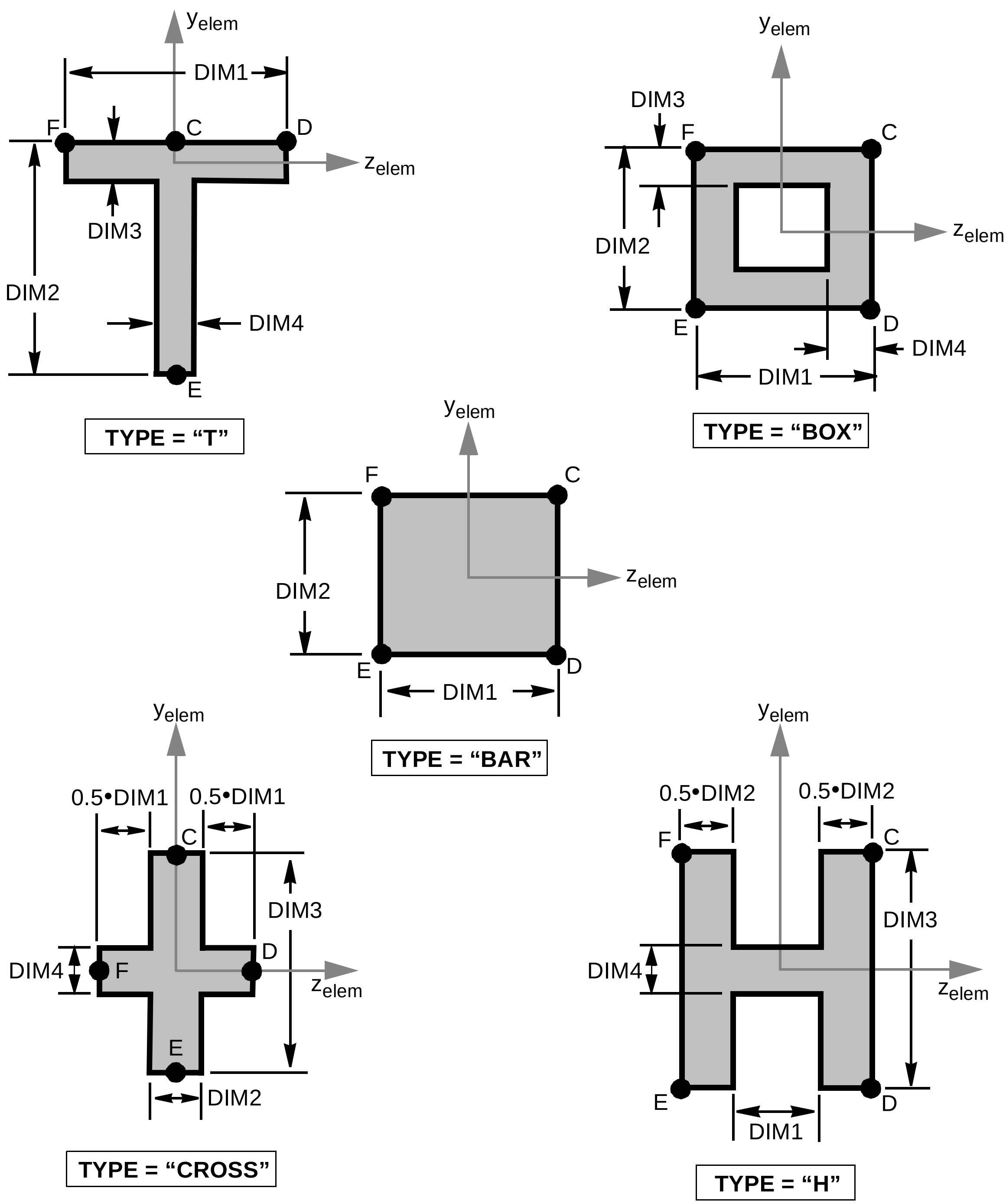
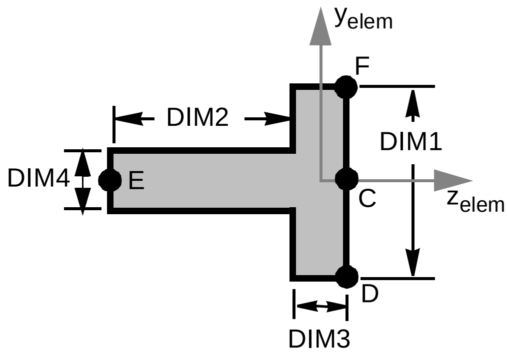


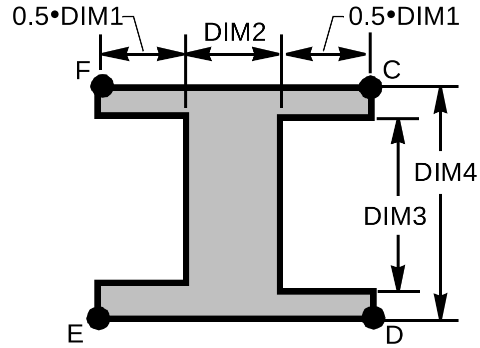
Figure 4-36 Dimensions and Data Recovery Locations for PBEAML Cross-Sections



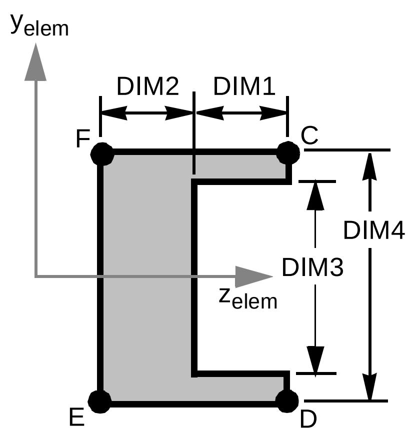
Dimensions and Data Recovery Locations for PBEAML Cross-Sections (continued)



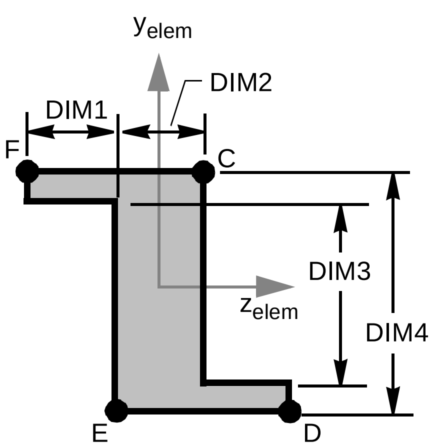
TYPE = "T1"



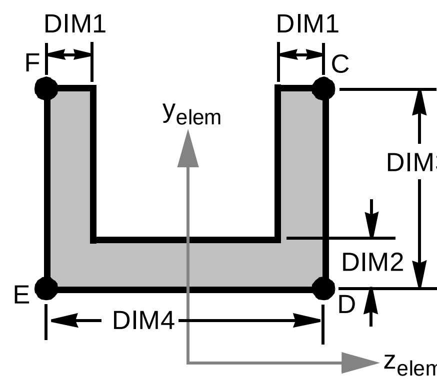
TYPE = "I1"



TYPE = "CHAN1"

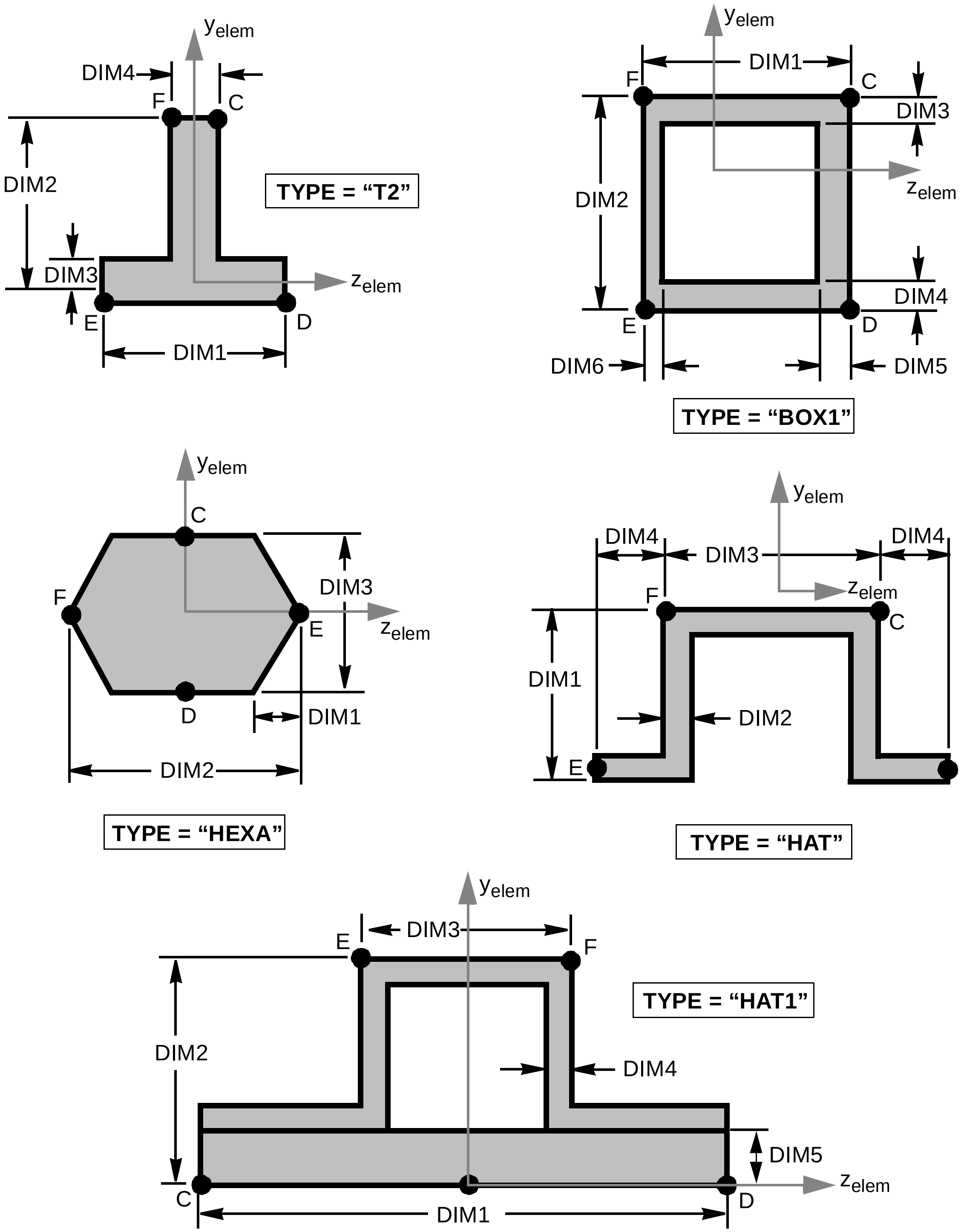


TYPE = "Z"



TYPE = "CHAN2"

Dimensions and Data Recovery Locations for PBEAML Cross-Sections (continued)



Dimensions and Data Recovery Locations for PBEAML Cross-Sections (continued)

PBUSH - *General Spring/Damper/Mass Properties*

Description

Defines the properties for general spring/damper/mass elements, defined via the CBUSH entry.

Format

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	“K”	K1	K2	K3	K4	K5	K6	
		“B”	B1	B2	B3	B4	B5	B6	
		“GE”	GE1	GE2	GE3	GE4	GE5	GE6	
		“M”	M11	M22	M33	I11	I22	I33	
		“M12”	I21	I23	I32				
		“RCV”	SA	ST	EA	ET			

Example

PBUSH	3	K	1000.		300.6		120.		
		B	8.	8.	12.				
		M	3.	3.	3.	124.	188.	56.	
		M12	123.	56.	76.				

Field	Description
PID	Property identification number. (Integer > 0)
“K”	Idicates that the next 6 fields contain stiffness terms (Character)
“B”	Indicates that the next 6 fields contain damping terms (Character)
“M”	Indicates that the next 6 fields contain mass/inertia terms (Character)
“M12”	Indicates that the next 3 fields contain coupling inertia terms (Character)
“GE”	Indicates that the next 6 fields contain structural damping coefficients (Character)
“RCV”	Indicates that the next 4 fields contain terms used in calculating stress or strain (Character)
Ki	Spring stiffness in the associated direction (in the element coordinate system) (Real, default = 0.0)
Bi	Damping values in the associated direction (in the element coordinate system) (Real, default = 0.0)
Mii	Mass values for the translational diagonal terms in the 6x6 element mass matrix (in the element coordinate system) (Real, default = 0.0)
Iii	Inertia values for the rotational diagonal terms in the 6x6 element mass matrix (in the element coordinate system) (Real, default = 0.0)
Iij	Inertia values for the off-diagonal rotational terms in the 6x6 element mass matrix (in the element coordinate system) (Real, default = 0.0)
GEi	Structural damping coefficient in the associated direction (in the element coordinate system) (Real, default = 0.0) See Remark 3
SA	Data recovery coefficients for element stress in the translational directions (Real, default =1.0) See Remark 5
ST	Data recovery coefficients for element stress in the rotational directions (Real, default =1.0) See Remark 5

Field	**Description**
EA	Data recovery coefficients for element strain in the translational directions (Real, default =1.0) See Remark 5
ET	Data recovery coefficients for element strain in the rotational directions (Real, default =1.0) See Remark 5

Remarks

1. PBUSH entries must have unique identification numbers with respect to all other property entries.
2. All values provided are in the element coordinate system..
3. Providing structural damping coefficients causes the calculation of an element structural damping matrix. The structural damping terms in the element coordinate system are equal to Ki*GEi. Therefore, in order to include structural damping, the associated Ki and GEi terms must both be non-zero. NOTE: If GE1 is provided and GE2-GE6 are blank, then GE2-GE6 will have the same value as GE1. If any of the values GE2-GE6 contain a value, then fields left blank default to 0.0.
4. The mass and inertia terms are similar to those of a CONM element. A concentrated mass with the 6x6 matrix defined by the terms is created at the element centroid in the element coordinate system.

$$
\begin{bmatrix}
M_{11} & 0 & 0 & 0 & 0 & 0 \\
0 & I_{22} & 0 & 0 & 0 & 0 \\
0 & 0 & I_{33} & 0 & 0 & 0 \\
0 & 0 & 0 & I_{11} & -I_{21} & -I_{31} \\
0 & 0 & 0 & -I_{21} & I_{22} & -I_{32} \\
0 & 0 & 0 & -I_{31} & -I_{32} & I_{33}
\end{bmatrix}
$$

5. The RCV terms are used to calculate element stress and strain output. Strain output is simply the element change in element length in the associated direction multiplied by the associated term. Stress output is the element force/moment in the associated direction multiplied by the associated term.
6. The value in field 3 of each line determines what the rest of the data on the line represents. There is no required order for the lines. That is, each line may have any one of the identifiers (“K”, “B”, “M”, “M12”, “GE”, “RCV”) on it. The only requirement is that only one line with each identifier is allowed on a single PBUSH entry.
7. If a PBUSH entry does not contain a line associated with one of the identifiers (“K”, “B”, “M”, “M12”, “GE”, “RCV”), the values associated with that identifier will have the default values.

PCOMP - *Layered Composite Element Property*

Description

Defines the properties of an n-ply laminated composite material.

Format

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	ZOC	NSM	SBOND	FT			LOPT	
	MID1	T1	TH1	SOUT1	MID2	T2	TH2	SOUT2	
	MID3	T3	TH3	SOUT3	-etc.-				

Example

PCOMP	100	-0.5	1.5	5.+3	HOFF			SYMMEM	
	150	0.05	90.	YES			-45.		
			45.0						

Field	Description
PID	Property identification number. (Integer > 0)
ZOC	Offset of the element reference plane (element bottom surface) from the plane of grid points.. (Real or blank)
NSM	Non-structural mass per unit area. (Real or blank)
SBOND	Allowable shear stress of the bonding material. (Real > 0.0 or blank. Required if FT is specified.)
FT	Failure theory. See Remark 5. (Character; one of the strings “HILL”, “HOFF”, “TSAI”, “STRESS”, “STRN”, or “STRAIN” or blank)
LOPT	Lamination generation option. See Remark 8. (Character; one of the strings “ALL”, “SYM”, “MEM”, or “SYMMEM” or blank. Default = “ALL”)
MIDi	Material identification number of the i-th layer. (Integer > 0 or blank)
Ti	Thickness of the i-th layer. (Real > 0.0 or blank)
THi	Angle between the longitudinal direction of the fibers of the i-th layer and the material X-axis. (Real or blank)
SOUTi	Stress output request for the i-th layer. (Character = “YES” or “NO”; Default = “NO”)

Remarks

1. PCOMP entries must have unique identification numbers with respect to all other property entries.
2. The plies are numbered from 1 to n beginning with the bottom layer.
3. Note that the offset, ZOC, is the distance from the plane of grid points to the bottom surface of the element. This is not the same as the offset, ZO, used with the CQUAD4 and CTRIA3 elements.
4. SBOND is required if bonding material failure index calculations are desired.
5. Fields 7 and 8 on line one are not currently used, but have been designated as containing real numbers. If values are enterred in these fields, they will not be used.
6. DIAG 40 will cause the listing of the generated Bulk Data entries.

7. The failure theory is used to determine the element failure on a ply-by-ply basis. The available theories are:

HILL	Hill Theory
HOFF	Hoffman Theory
TSAI	Tsai-Wu Theory
STRESS	Maximum Stress Theory
STRAIN	Maximum Strain Theory

8. To minimize input requirements several lamination options (LOPT) are available. “ALL” indicates that every ply is specified. “SYM” indicates that ply layup is symmetric about the center ply and that the plies on one side of the center line are specified. “SYMMEM” indicates a symmetric layup of membrane only plies.
9. The material properties, MIDi, may only reference MAT1, MAT2, or MAT8 Bulk Data entries.
10. If any of MIDi, Ti, or THi are blank, then the last non-blank values specified for each will be used to define the values for the ply.

PCOMP1 - *Layered Composite Element Property*

Description

Defines the properties of an n-ply laminated composite material where all plies are composed of the same material and are of equal thickness.

Format

1	2	3	4	5	6	7	8	9	10
PCOMP1	PID	ZOC	NSM	SBOND	FT	MID	TPLY	LOPT	
	TH1	TH2	TH3	-etc.-					

Example

PCOMP1	100	-0.5	1.7	5.+3	STRAIN	200	0.25	SYM	
	-45.0	45.0	90.0	90.0	45.0				

Field	Description
PID	Property identification number. (Integer > 0)
ZOC	Offset of the element reference plane (element bottom surface) from the plane of grid points. (Real or blank)
NSM	Non-structural mass per unit area. (Real or blank)
SBOND	Allowable shear stress of the bonding material. (Real > 0.0)
FT	Failure theory. See Remark 4. (Character; one of the strings "HILL", "HOFF", "TSAI", "STRESS", or "STRAIN")
MID	Material identification number for all layers. (Integer > 0)
LOPT	Lamination generation option. See Remark 5. (Character; one of the strings "ALL", "SYM", "MEM", or "SYMMEM")
TPLY	Thickness of all layers. (Real > 0.0 or blank)
THi	Angle between the longitudinal direction of the fibers of the ith layer and the material X-axis. (Real or blank)

Remarks

1. PCOMP1 entries must have unique identification numbers with respect to all other property entries.
2. The plies are numbered from 1 to n beginning with the bottom layer.
3. The offset (ZOC) is not the same offset (ZO) used in the CQUAD4 and CTRIA3 entries. ZOC references the bottom surface of the element.
4. SBOND is required if bonding material failure index calculations are desired.
5. The failure theory is used to determine the element failure on a ply-by-ply basis. The available theories are:

HILL	Hill Theory
HOFF	Hoffman Theory
TSAI	Tsai-Wu Theory
STRESS	Maximum Stress Theory
STRAIN	Maximum Strain Theory

6. To minimize input requirements several lamination options (LOPT) are available. ALL indicates that every ply is specified. SYM indicates that ply layup is symmetric about the center ply and that the plies on one side of the center line are specified. SYMMEM indicates a symmetric layup of membrane only plies.
7. The material property, MID, may reference only MAT1, MAT2, and MAT8 Bulk Data entries.

PCOMP2 - *Layered Composite Element Property*

Description

Defines the properties of an n-ply laminated composite material where all plies are composed of the same material.

Format

1	2	3	4	5	6	7	8	9	10
PCOMP2	PID	ZOC	NSM	SBOND	FT	MID		LOPT	
	T1	TH1	T2	TH2	-etc.-				

Example

PCOMP2	100	-0.5	1.7	5.+3	TSAI	200		SYM	
	0.25	-45.0	0.5	90.0	0.25	45.0			

Field	Description
PID	Property identification number. (Integer > 0)
ZOC	Offset of the element reference plane (element bottom surface) from the plane of grid points. (Real or blank)
NSM	Non-structural mass per unit area. (Real or blank)
SBOND	Allowable shear stress of the bonding material. (Real > 0.0)
FT	Failure theory. See Remark 4. (Character; one of the strings, “HILL”, “HOFF”, “TSAI”, “STRESS”, or “STRAIN”)
MID	Material identification number for all layers. (Integer > 0 or blank)
LOPT	Lamination generation option. See Remark 5. (Character; one of the strings, “ALL”, “SYM”, “MEM”, or “SYMMEM”)
Ti	Thickness of the ith layer. (Real > 0.0 or blank)
THi	Angle between the longitudinal direction of the fibers of the ith layer and the material X-axis. (Real or blank)

Remarks

1. PCOMP2 entries must have unique identification numbers with respect to all other property entries.
2. The plies are numbered from 1 to n beginning with the bottom layer.
3. The offset (ZOC) is not the same offset (ZO) used in CQUAD4 and CTRIA3 entries. ZOC references to the bottom surface of the element.
4. SBOND is required if bonding material failure index calculations are desired.
5. The failure theory is used to determine the element failure on a ply-by-ply basis. The available theories are:

HILL	Hill Theory
HOFF	Hoffman Theory
TSAI	Tsai-Wu Theory
STRESS	Maximum Stress Theory
STRAIN	Maximum Strain Theory

6. To minimize input requirements several lamination options (LOPT) are available. ALL indicates that every ply is specified. SYM indicates that ply layup is symmetric about the center ply and that the plies on one side of the center line are specified. SYMMEM indicates a symmetric layup of membrane only plies.

7. The material property, MID, may reference only MAT1, MAT2, and MAT8 Bulk Data entries.
8. If any of the Ti or THi are blank, then the last non-blank values specified for each will be used to define the values for the ply.

PCONEAX - *Conical Shell Element Properties*

Description

Defines the properties of a conical shell element described on a CCONEAX entry.

Format

1	2	3	4	5	6	7	8	9	10
PCONEAX	ID	MID1	T1	MID2	I	MID3	T2	NSM	
	Z1	Z2	PHI1	PHI2	PHI3	PHI4	PHI5	PHI6	
	PHI7	PHI8	PHI9	PHI10	PHI11	PHI12	PHI13	PHI14	

Example

PCONEAX	2	4	1.0	6	16.3	8	2.1	0.5	
	0.001	-0.002	23.6	42.9					

Field	Description
ID	Property identification number. (Integer > 0)
MIDi	Material identification numbers for membrane, bending, and transverse shear, respectively. (Integer ≥ 0)
T1, T2	Membrane thickness and transverse shear thickness. (Real > 0.0 if MIDi ≠ 0)
I	Moment of inertia per unit width. (Real)
NSM	Nonstructural mass per unit area. (Real)
Z1, Z2	Fiber distances for stress recovery. (Real)
PHIi	Azimuthal coordinates (in degrees) for stress recovery. (Real)

Remarks

1. PCONEAX entries must have unique identification numbers with respect to all other property entries.
2. This entry is allowed only if an AXIC entry is also present.
3. PCONEAX entries may only reference MAT1 material entries.
4. .If either MID1 = 0 or blank or T1 = 0.0 or blank, then both must be zero or blank.
5. .If either MID2 = 0 or blank or I = 0.0 or blank, then both must be zero or blank.
6. .If either MID3 = 0 or blank or T2 = 0.0 or blank, then both must be zero or blank.
7. A maximum of 14 azimuthal coordinates for stress recovery may be specified on up to two continuation entries.
8. .For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.

PDAMP - *Scalar Damper Property*

Description

Used to define the damping value of a CDAMP1 or CDAMP3 scalar damper element.

Format

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

Example

PDAMP	14	-2.3	2	6.1					

Field	Description
PIDi	Property identification number. (Integer > 0)
Bi	Value of scalar damper. (Real)

Remarks

1. PDAMP entries must have unique identification numbers with respect to all other property entries.
2. Up to four damping coefficients may be defined on a single entry.
3. CDAMP2 and CDAMP4 entries may be used to define damping elements without reference to a PDAMP entry. A structural viscous damper, CVISC, may also be used for geometric grid points
4. Though allowed, caution should be exercised when using negative damping coefficients.
5. For a discussion of scalar elements, see Section 5.6 of the Theoretical Manual.

PDUMi - *User-Defined Element Property*

Description

Defines the properties of a user-defined (dummy) element ($1 \leq i \leq 9$). Referenced by the CDUMi entry.

Format

1	2	3	4	5	6	7	8	9	10
PDUMi	PID	MID	A1	A2	A3	A4	A5	A6	
	A7	-etc.-							

Example

PDUMi	108	2	2.4	9.6	1.E4	15.		3.5	
	5		2						

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer >)
Ai	Additional fields. (Real or Integer)

Remarks

1. PDUMi entries must have unique identification numbers with respect to all other property entries.
2. The additional fields are defined by the user-written element routines.

PELAS - *Scalar Elastic Properties*

Description

Used to define the stiffness, damping coefficient, and stress coefficient of CELAS1 and CELAS3 scalar elastic (spring) elements.

Format

1	2	3	4	5	6	7	8	9	10
PELAS	PID1	K1	GE1	S1	PID2	K2	GE2	S2	

Example

PELAS	7	4.29	0.06	7.92	27	2.17	0.0032		

Field	Description
PIDi	Property identification number. (Integer > 0)
Ki	Spring stiffness. (Real)
GEi	Damping coefficient. (Real)
Si	Stress coefficient. (Real)

Remarks

1. PELAS entries must have unique identification numbers with respect to all other property entries.
2. Up to two sets of elastic spring properties may be defined on a single entry.
3. CELAS2 and CELAS4 entries may be used to define scalar spring elements without reference to a PELAS entry
4. Though allowed, caution should be exercised when using negative spring values.
5. For a discussion of scalar elements, see Section 5.6 of the Theoretical Manual.

PELBOW - *Curved Beam Properties*

Description

Defines the properties of a curved beam, defined via the CELBOW entry.

Format

1	2	3	4	5	6	7	8	9	10
PELBOW	PID	MID	A	I1	I2	J	NSM		
	R1	THETA1	R2	THETA2	R3	THETA3	R4	THETA4	
	K1	K2	C	KX	KY	KZ	R	BETA	

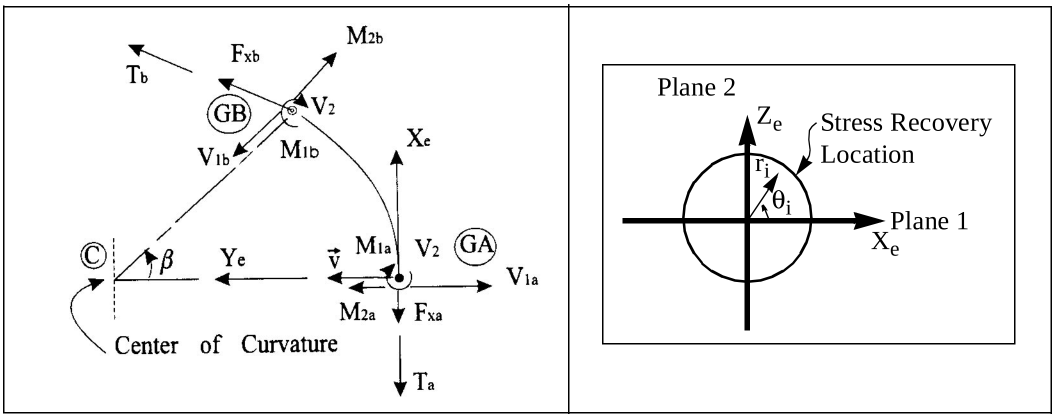
Example

PELBOW	2	6061	16.0	211.0	211.0	422.0	6.0		
	5.3	0.0	5.3	90.0	5.3	180.0	5.3	270.0	
	2.0	2.0	1.0	1.0	5.76	5.76	15.0	90.0	

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
A	Area of cross section. (Real > 0.0)
I1	Area moment of inertia in Plane 1. (Real > 0.0))
I2	Area moment of inertia in Plane 2. (Real > 0.0)
J	Torsional constant. (Real > 0.0)
NSM	Nonstructural mass per unit length. (Real)
Ri, THETAi	Stress recovery coefficients. See Remark 5. (Real, THETAi in degrees)
K1, K2	Area factors for shear. (Real)
C	Stress intensification factor. (Real)
KX, KY, KZ	Flexibility correction factors. (Real)
R	Radius of curvature of the element. (Real > 0.0)
BETA	Angle, in degrees, from GA to GB. See Remark 5. (Real, 0. < BETA < 180.)

Remarks

1. PELBOW entries must have unique identification numbers with respect to all other property entries.
2. For structural problems, PELBOW may only reference MAT1 entries.
3. .The product moment of inertia is zero (I12 = 0). This assumes that at least one axis of symmetry of the element cross section exists, for example, tube, I-beam, channel, tee, etc.
4. Certain PELBOW properties (e.g., stress recovery locations) are defined in terms of the CELBOW element coordinate system, as shown in **Figure 4-37**.



Element Local Coordinate System | Element Cross-Section

Figure 4-37 CELBOW Element Coordinate System

PGAP - *GAP property definition*

Description

Defines the properties for a GAP.

Format

1	2	3	4	5	6	7	8	9	10
PGAP	PID	U0	F0	KA	KB	KT	MU1	MU2	
	TMAX	MAR	TRMIN						

Example

PGAP	100	.01	200.	1000.	4.97				

Field	Description
PID	Property identification number. (Integer > 0)
U0	Initial GAP opening (Real, default = 0.0)
F0KA,KB,KT, MU1,MU2, TMAX,MAR, TRMIN	Not used (Real, default = 0.0)

Remarks

1. PGAP entries and their associated CGAP entries are converted into equivalent GAP data in the program.
2. See the GAP entry for more information on the GAP constraint.
3. The GAP constraint is used to measure openings between two GRID points, or a GRID point and ground. In SOL's 1 and 2, it may be used as an interative GAP constraint.
4. In transient response, NOLINi entries may ne used to model a GAP which opens and closes.

PLFACT - *Piecewise Linear Analysis Factor Definition*

Description

Defines scale factors for piecewise linear analysis loading.

Format

1	2	3	4	5	6	7	8	9	10
PLFACT	SID	B1	B2	B3	B4	B5	B6	B7	
	B8	B9	etc.						

Example

PLFACT	6	0.2	0.3	0.4	0.5	0.6	0.7	0.8	
	0.9	1.0							

Field	Description
SID	Unique set identification number. (Integer > 0)
Bi	Loading factor. (Real)

Remarks

1. At any stage of the piecewise linear analysis, the accumulated load is given by: $\{P_i\} = B_i \{P\}$ where {P} is the total load defined via static load entries. For example, to load the structure in ten equally spaced load increments, Bi = 0.1 * i ; i = 1, 10.
2. Normally, the Bi form a monotonically increasing sequence. A singular stiffness matrix will result if Bi = B(i-1).
3. At least two factors must be defined.
4. Piecewise linear analysis factor sets must be selected in Case Control (PLCOEFF = SID) to be used by NASTRAN-CORE.

PLOAD - *Shell Element or Solid Element Face Pressure Load*

Description

Defines a static pressure load on a two-dimensional shell element or on a face of a three-dimensional solid.

Format

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			

Example

PLOAD	1	-4.0	16	32	11				

Field	Description
SID	Load set identification number. (Integer > 0)
P	Pressure value in units of force per unit area. (Real)
Gi	Grid point identification number. (Integer > 0; G4 may be left blank depending on element type)

Remarks

1. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. Grid points must be unique and non-colinear.
4. The direction of positive pressure is determined by the grid point ordering provided on this entry, and acts normal to the surface. The convention for triangular shell elements and solid element faces are shown in **Figure 4-38**. Pressure loads are applied as equivalent concentrated loads, simply with one-third of the total force (pressure*area) acting at each grid, in a direction normal to the surface. No other considerations are given to element geometry.

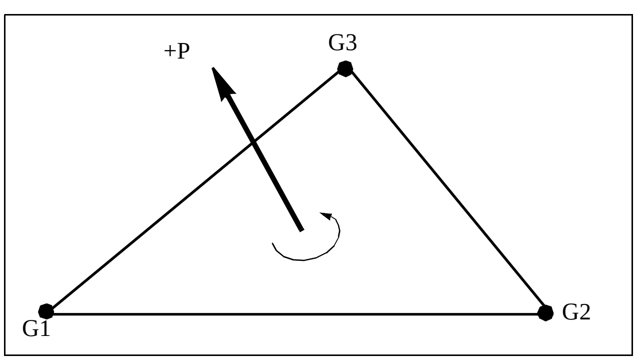


Figure 4-38 Triangular Element and Solid Element Face Pressure Conventions

5. Pressure loading conventions for quadrilateral shell elements and solid element faces are shown in **Figure 4-39**. As with triangular elements and faces, pressure loads are applied as equivalent concentrated loads at the grid points, in a direction normal to the face. Since the surface defined by the four grid points is not necessarily planar, approximate grid point loads are computed as follows:
   - A pair of triangular elements is formed by bisecting the quadrilateral by one of its diagonals. This is done twice, to form two pairs of overlapping triangular elements.
   - One-fourth of the resulting load (pressure*area) is applied to each triangular element, and the grid point forces computed as described in Remark 3. The summation of these triangular contributions results in the total load.

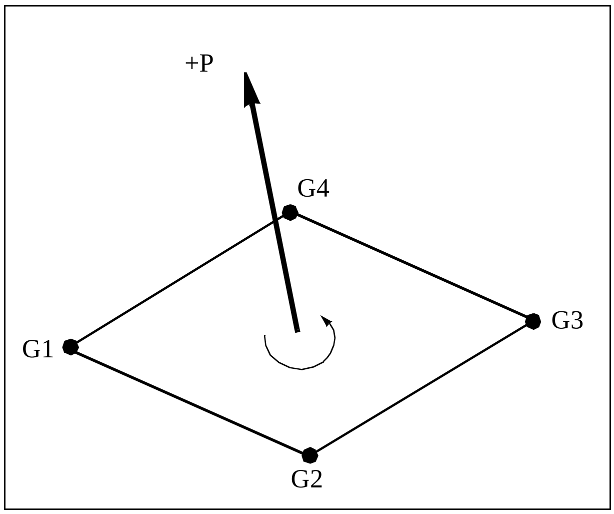


Figure 4-39 Quadrilateral Element and Solid Element Face Pressure Conventions

PLOAD1 - *Applied Load on CBAR, CBEAM,or CELBOW Elements*

Description

Defines concentrated, uniformly distributed, or linearly distributed applied loads to CBAR or CBEAM elements at user-selected points or intervals along the element. May also be used to define distributed loads along the entire length of CELBOW elements.

Format

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

Example

PLOAD1	25	1065	MY	FR	0.2	2.5E3	0.8	3.5E3	

Field	Description
SID	Load set identification number. (Integer > 0)
EID	CBAR or CELBOW element identification number. (Integer > 0)
TYPE	Load type. See Remark 5 for allowable types. (Character)
SCALE	Determines basis (scale) in which X1, X2 distances are expressed. See Remark 6 for allowable types. (Character)
Xi	Distances along the CBAR element from end A. (Real; $0 \leq X1 \leq X2$; X2 may also be blank)
Pi	Load factors at Xi positions. (Real or blank)

Remarks

1. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. If X2 is blank or equal to X1, a concentrated load of value P1 will be applied at the X1 position.
4. If $X2 \neq X1$, a linearly varying distributed load will be applied to the element between positions X1 and X2, with an intensity per unit length equal to P1 at X1 and P2 at X2. Lengths may either be actual or projected, as noted in Remark 5.
5. The TYPE field is used to define loads according to the following character strings:
   - "FX", "FY", or "FZ": Forces in the x, y, or z direction of the basic coordinate system.
   - "MX", "MY", or "MZ": Moments in the x, y, or z direction of the basic coordinate system.
   - "FXE", "FYE", or "FZE": Forces in the x, y, or z direction of the local element coordinate system.
   - "MXE", "MYE", or "MZE": Moments in the x, y, or z direction of the local element coordinate system.
6. The SCALE field may assume any of the following character strings, indicating which basis of measurement is to be used:
   - "LE": Length. The Xi values are the actual distances along the element's axis and, if $X1 \neq X2$, the Pi are load intensities per unit length of the element.
   - "FR": Fractional. The Xi values are ratios (<1.0) of the distance along the x axis to the total length and, if $X1 \neq X2$, the Pi are load intensities per unit length of the element.
7. For the CELBOW element, the following conventions and restrictions apply:

- The element x and y axes are the R and θ directions, respectively.
- Only distributed loads over the entire length of the CELBOW element may be applied (X1 and X2 are ignored.)
- Projected loads are not applicable.

8. Resulting CBEAM element loads are applied along the line of shear center.
9. If element stress and/or force output is requested, output will be provided at the Xi locations in addition to output normally provided at the element end points.
10. If any of the element coordinate TYPE representations are used (e.g., "FXE"), only actual, and not projected SCALE options may be used .
11. The following example illustrates a typical PLOAD1 application. Note the following:
    - If SCALE = "LE", the total load applied to the bar is $\left(\frac{P1 + P2}{2}\right) \cdot (X2 - X1)$ in the y-basic direction.

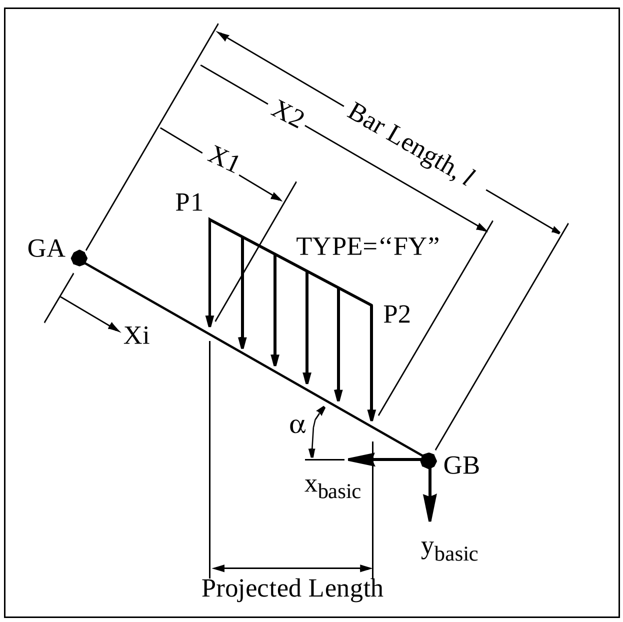


Figure 4-40 PLOAD1 CBEAM Element Loading Example

PLOAD2 - *Shell Element Pressure Load*

Description

Defines a uniform static pressure load on the CQUAD4, CSHEAR, or CTRIA3 shell elements.

Format

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	-etc.-							

Example

PLOAD2	21	-3.6	1	4	16	19	21		

Alternate Format and Example:

PLOAD2	SID	P	EID1	EID2	EID3	“THRU”	EID4	EID5	
	EID6	-etc.-							
PLOAD2	21	-3.6	1	4	16	THRU	22	98	
	127								

Field	Description
SID	Load set identification number. (Integer > 0)
P	Pressure value in units of force per unit area. (Real)
EIDi	Element identification numbers. (Integer > 0)

Remarks

1. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. The pressure load is computed for each element as if the grid points to which the element is connected were specified on a PLOAD entry. The grid point sequence specified on the element connection entry is assumed for the purpose of computing pressure loads.
4. All elements referenced must exist.
5. Embedded blank fields are not allowed.
6. PLOAD2 entries are converted (internally) into PLOAD4 entries for processing.

PLOAD4 - *Pressure Loads on Face of Structural Elements*

Description

Defines pressure loading on CTRIA3, CQUAD4, CHEXA, CTETRA, or CPENTA element faces.

Format

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3/G4	
	CID	N1	N2	N3					

Example

PLOAD4	101	2043	15.	18.	23.6		51	62	
	52	1.0	0.	0.					

Alternate Format and Example (See Remark 6):

PLOAD4	SID	EID1	P1	P2	P3	P4	“THRU”	EID2	
	CID	N1	N2	N3					

PLOAD4	1001	452	105.				THRU	568	
	2375	0.	1.	1.					

Field	Description
SID	Load set identification number. (Integer > 0)
EIDi	Element identification number. (Integer > 0; for THRU option, EID1 < EID2)
Pi	Pressure values at grid points, in units of force per unit area. P1 must be specified. See Remarks 2, 3, 4, and 5. (Real or blank; Default for P2, P3, P4 is P1)
CID	Coordinate system identification number. (Integer ≥ 0 or blank; Default = 0)
Gi	Grid point identification number. See Remark 4. (Integer > 0 or blank)
Ni	Components of a vector in system CID that defines the direction (but not the magnitude) of the pressure. (Real or blank)

Remarks

1. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. The continuation entry is optional. If not supplied, the direction of positive pressures are:
   - inward, and normal to the face, for CTETRA, CPENTA and CHEXA elements
   - normal to the surface and determined using the right-hand rule applied to the grid connectivities for plate and shell elements
4. If P2, P3, and P4 are blank, the loading intensity is uniform and equal to P1.
5. Pi and Gi conventions for the various element types are as follows:

- CTETRA: G1 is a grid point on the face to be loaded, and G4 is the grid point not on the face (note that this is unique.) Pressures P1 through P3 are the pressures at each grid point on the face, with ordering determined by applying the right hand rule about an outward normal vector.
- CPENTA: For CPENTA's triangular faces, G1 is a point on the surface, and the G3/G4 field is simply left blank. For the quadrilateral faces, G1 is a point on the surface to be loaded, and G3 is the point on the face diagonally opposite from G1. P1 through P3 (and perhaps P4) are supplied as required, with ordering determined from the right hand rule applied to an outward normal vector.
- CHEXA: G1 is a point on the surface to be loaded, and G3 is the point on the face diagonally opposite from G1. P1 through P3 (and perhaps P4) are supplied as required, with ordering determined from the right hand rule applied to an outward normal vector
- Plate and shell elements: G1 and G3/G4 are ignored, P1 through P3 (and perhaps P4) are supplied as required. P1 is applied to the first grid point listed on the element connection entry, P2 the second, and so on.

6. Note, too, the following with respect to pressure loads:
   - Pressures are given in force per unit area of the element face, not in terms of projected unit area.
   - If a direction vector has not been specified and the element surface is not planar, the direction of the pressure will vary over the surface
   - A uniform pressure may not result in equal grid point loads.
7. The alternate entry format is only available for plate and shell elements.

PLOTEL - *Fictitious Plot Element Definition*

Description

Defines a fictitious one-dimensional element for use in plotting.

Format

1	2	3	4	5	6	7	8	9	10
PLOTEL	EIDA	G1A	G2A		EIDB	G1B	G2B		

Example

1	2	3	4	5	6	7	8	9	10
PLOTEL	29	35	16						

Field	Description
EIDi	Element identification number. (Integer > 0)
G1i, G2i	Grid point identification numbers of connection points. (Integer > 0, G1i, G2i unique)

Remarks

1. This element is not used in the model during any of the solution phases of a problem, and is only used to simplify plotting of structures with large numbers of colinear grid points where the plotting of each one along with the elements connecting them would result in a confusing plot.
2. Each element identification number must be unique with respect to all other element identification numbers.
3. Up to two PLOTEL elements may be defined on a single entry.

PMASS - *Scalar Mass Properties*

Description

Defines the mass of scalar mass elements defined on CMASS1 or CMASS3 entries.

Format

1	2	3	4	5	6	7	8	9	10
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

Example

PMASS	7	4.29	6	13.2					

Field	Description
PIDi	Property identification number. (Integer > 0 or blank)
Mi	Scalar mass value. (Real or blank)

Remarks

1. PMASS entries must have unique identification numbers with respect to all other property entries.
2. From one to four mass properties may be defined on a single PMASS entry.
3. The CMASS2 and CMASS4 entries provide an alternate method of scalar mass definition by providing mass values directly, without reference to a PMASS entry.
4. Though negative masses may be defined, caution should be used when doing so.
5. For a discussion of scalar elements, see Section 5.6 of the Theoretical Manual.

POINTAX - *Axisymmetric Point*

Description

Defines the location of a point on an axisymmetric ring at which displacement output may be requested and loads applied via the FORCE, FORCEAX, MOMENT, or MOMAX entries.

Format

1	2	3	4	5	6	7	8	9	10
POINTAX	ID	RID	PHI						

Example

POINTAX	2	3	30.0						

Field	Description
ID	Point identification number. See Remark 2. (Integer > 0)
RID	Identification number of a RINGAX entry. (Integer > 0)
PHI	Azimuthal angle in degrees. (Real)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Each POINTAX identification number must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
3. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.
4. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
5. For a discussion of the axisymmetric solid problem, see Section 5.11 of the Theoretical Manual.

PRESAX - *Axisymmetric Pressure Load*

Description

Defines the static pressure loading for a model containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
PRESAX	SID	P	RID1	RID2	PHI1	PHI2			

Example

PRESAX	3	7.92	4	3	20.6	31.4			

Field	Description
SID	Load set identification number. (Integer > 0)
P	Pressure value. (Real)
RID1, RID2	Ring identification numbers, defined on RINGAX entries. (Integer > 0)
PHI1, PHI2	Azimuthal angles in degrees. (Real, PHI1 ≠ PHI2)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Load sets must be selected in Case Control (LOAD = SID) in order to be used by NASTRAN-CORE.
3. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
4. For a discussion of the axisymmetric solid problem, see Section 5.11 of the Theoretical Manual.

PROD - *Rod Element Properties*

Description

Defines CROD element properties.

Format

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

Example

PROD	17	23	42.6	17.92	4.236	0.5			

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
A	Area of rod. (Real)
J	Torsional constant. (Real)
C	Coefficient to determine torsional stress. (Real)
NSM	Nonstructural mass per unit length. (Real)

Remarks

1. PROD entries must have unique identification numbers with respect to all other property entries.
2. For structural problems, PROD entries may only reference MAT1 material entries.

PSHEAR - *Shear Panel Property*

Description

Defines the elastic properties of a shear panel. Referenced by the CSHEAR entry.

Format

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID1	MID1	T1	NSM1	F1	F2			

Example

PSHEAR	13	2	4.9	16.2					

Field	Description
PIDi	Property identification number. (Integer > 0)
MIDi	Material identification number. (Integer > 0)
Ti	Thickness of shear panel. (Real ≠ 0.0)
NSMi	Nonstructural mass per unit area. (Real)
F1	Factor used to create equivalent ROD element along edges 1-2 and 3-4. See remark 3. (Real ≥ 0., default = 0.0)
F2	Factor used to create equivalent ROD element along edges 2-3 and 1-4. See remark 3. (Real ≥ 0., default = 0.0)

Remarks

1. PSHEAR entries must have unique identification numbers with respect to all other property entries.
2. PSHEAR entries may only reference MAT1 material entries.
3. The stiffnes equavalent to axial-force only ROD elements can be created on the edges of the element by using factors F1 and F2.If the associated factor is less than 1.01, then the area used for the equivalent ROD elements is set equal to $0.5*F_i*T*W$.where F_i is the associated factor, W is the average width of the panel. If the associated factor is greater than 1.01, then the area used for the equivalent ROD is $.5*F_i*T$.

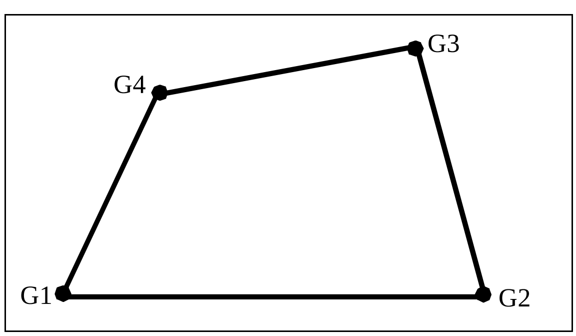


Figure 4-41 Shear Panel Extensional Area

PSHELL - *Shell Element Properties*

Description

Defines the membrane, bending, transverse shear, and coupling properties of the CQUAD4 and CTRIA3 shell elements.

Format

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T**3	MID3	TS/T	NSM	
	Z1	Z2	MID4	THETA or MCID	SCID	ZOFFS			

Example

PSHELL	203	204	1.90	205	1.2	206	0.8	6.32	
	+0.95	-0.95		0	0	0.01			

Field	Description
PID	Property identification number. (Integer > 0)
MID1	Material identification number for membrane effects. (Integer > 0 or blank)
T	Default value for membrane thickness. (Real > 0.0)
MID2	Material identification number for bending. (Integer > 0 or blank)
12I/T**3	Bending moment of inertia ratio, $(12I)/T^3$. (Real or blank; Default = 1.0)
MID3	Material identification number for transverse shear. (Blank unless MID2 > 0; Integer > 0 otherwise)
TS/T	Transverse shear thickness ratio, T_s/T. Ratio of the shear thickness, T_s, to the membrane thickness of the shell, T. (Real > 0.0 or blank; Default = 0.833333)
NSM	Nonstructural mass per unit area. (Real or blank)
Z1, Z2	Fiber distances for stress computation. The positive direction is determined by applying the right hand rule to counterclockwise ordering of the grid points on the connection entry. (Real or blank; Defaults are -T/2 for Z1 and +T/2 for Z2)
MID4	Material identification number for membrane-bending coupling. (Blank unless MID1 > 0 and MID2 > 0; Integer > 0 otherwise. MID4 not equal to MID1 or MID2.
THETA	Material property orientation specification, given as a real number in degrees. See Remark 8. (Real or blank; Default = 0.0)
MCID	Coordinate system ID for material property orientation specification. The element material x-axis is determined by the projection of the MCID x-axis onto the plane of the element. See Remark 8. (Integer ≥ 0 or blank; Default, THETA = 0.0)
SCID	Identification number of stress coordinate system. See Remark 9. (Real or blank, or Integer > 0)
ZOFFS	Element reference plane (element mid-plane) offset from the plane of grid points. See Remark 8. (Real ≥ 0.0 or blank; Default = 0.0)

Remarks

1. PSHELL entries must have unique identification numbers with respect to all other property entries.

2. The results of leaving any MIDi field blank are:

MID1	No membrane or coupling stiffness, mass, or structural damping
MID2	No bending, coupling, or transverse shear stiffness
MID3	No transverse shear flexibility
MID4	No membrane-bending coupling

3. The continuation entry is not required.
4. Structural damping is computed based on MID1 reference to a MATi entry with nonzero GE field.
5. The MID4 field should be left blank if the material properties are symmetric with the mid-plane of the shell.
6. For structural problems, PSHELL entries may reference MAT1, MAT2, or MAT8 material property entries.
7. If the transverse shear material, MID3, references a MAT2 entry then G33 must be zero. If MID3 references MAT8 data, then G1Z and G2Z must not be zero.
8. Nonblank THETA/MCID, and ZOFFS fields provide defaults for corresponding blank fields on the CQUAD4 and CTRIA3 bulk data entries.
9. The stress results for any element referencing a PSHELL will be reported in the SCID. The default value for SCID is blank, which means the element coordinate system. If SCID is an integer, it is the id of a CORDxx entry and the stress will be reported in the referenced coordinate system. A value of 0 for SCID will get the element stress results in the BASIC coordinate system. A real number for SCID is an angle. If a real value is provided, the stress output for the element will be reported in a coordinate system which has been rotated SCID degrees about the element Z-axis from the element coordinate system.
10. If a value is provided for TS/T, then MID3 must be specified, otherwise a FATAL error will occur.
11. If MID1 points to a MAT2 entry, then both G11 and G22 on that MAT2 must be non-zero. If for some reason, you feel that this is overly restrictive, you can use SYSTEM cell 126 to allow zero values.on the MAT2, however, the element will not be stable for membrane loads by itself.

PSOLID - *Solid Element Properties*

Description

Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries.)

Format

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM	IN					

Example

PSOLID	15	3							

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
CORDM	Material coordinate system
IN	Select integration scheme used (integer, default = 0)

Remarks

1. PSOLID entries must have unique identification numbers with respect to all other property entries.
2. MID must reference a MAT1 (isotropic) or MAT9 (anisotropic) entry.
3. Element stress output for HEXA and PENTA elements will be in the material coordinate system.
4. The material coordinate system provides the orientation for the material constants provided on a MAT9 entry
5. The default integration scheme for the HEXA, PENTA, and TETRA elements is reduced integration. A non-zero value for IN will result in full integration being used. Full integration may result in overly stiff elements and is not recommended.
6. If the default is used for IN, the element stiffness is calculated using reduced stiffness, but the mass is calculated using full integration. In most cases, reduced integration will provide more accurate results.

PTORDRG - *Toroidal Ring Properties*

Description

Used to define membrane and flexure (bending) properties of a toroidal ring element. Referenced by a CTORDRG entry.

Format

1	2	3	4	5	6	7	8	9	10
PTORDRG	PID	MID	TM	TF	PID	MID	TM	TF	

Example

1	2	3	4	5	6	7	8	9	10
PTORDRG	2	4	0.1	0.15					

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
TM	Thickness for membrane. (Real > 0.0)
TF	Thickness for flexure. (Real)

Remarks

1. PTORDRG entries must have unique identification numbers with respect to all other property entries.
2. The material identification number MID may only reference a MAT1 or MAT3 entry.
3. .Up to two sets of toroidal ring properties may be defined on a single entry.

PTRAPAX - *Triangular Ring Element Properties*

Description

Defines the properties of an axisymmetric trapezoidal cross-section ring element defined on a CTRAPAX entry.

Format

1	2	3	4	5	6	7	8	9	10
PTRAPAX	PID		MID	PHI1	PHI2	PHI3	PHI4	PHI5	
	PHI6	PHI7	PHI8	PHI9	PHI10	PHI11	PHI12	PHI13	
	PHI14								

Example

PTRAPAX	5		15	0.0	5.0	6.0	7.0	8.0	
	9.0	10.0	15.0	20.0	25.0	30.0	35.0	40.0	
	45.0								

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
PHIi	Azimuthal coordinates, in degrees, for stress recovery. (Real)

Remarks

1. PTRAPAX entries must have unique identification numbers with respect to all other property entries.
2. This entry is only allowed if an AXIC entry is also present.
3. PTRAPAX entries may reference MAT1 or MAT3 material entries.
4. A maximum of 14 azimuthal coordinates for stress recovery may be specified on this entry.

PTRIAAX - *Triangular Ring Element Properties*

Description

Defines the properties of an axisymmetric triangular cross-section ring element defined on a CTRIAAX entry.

Format

1N1	2	3	4	5	6	7	8	9	10
PTRIAAX	PID		MID	PHI1	PHI2	PHI3	PHI4	PHI5	
	PHI6	PHI7	PHI8	PHI9	PHI10	PHI11	PHI12	PHI13	
	PHI14								

Example

PTRIAAX	5		15	0.0	5.0	6.0	7.0	8.0	
	9.0	10.0	15.0	20.0	25.0	30.0	35.0	40.0	
	45.0								

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer >0)
PHIi	Azimuthal coordinates, in degrees, for stress recovery. (Real)

Remarks

1. PTRIAAX entries must have unique identification numbers with respect to all other property entries.
2. This entry is only allowed if an AXIC entry is also present.
3. PTRIAAX property entries may only reference MAT1 or MAT3 material entries.
4. A maximum of 14 azimuthal coordinates for stress recovery may be specified on this entry.

PTUBE - *Tube Property*

Description

Defines the properties of a thin-walled cylindrical tube element. Referenced by the CTUBE entry.

Format

1	2	3	4	5	6	7	8	9	10
PTUBE	PID	MID	OD	T	NSM				

Example

PTUBE	2	6	6.29	0.25					

Field	Description
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
OD	Outside diameter of tube. (Real > 0.0)
T	Thickness of tube. (Real; T ≤ 1/2 OD)
NSM	Nonstructural mass per unit length. (Real; Default = 0)

Remarks

1. PTUBE entries must have unique identification numbers with respect to all other property entries.
2. If T is zero, a solid circular rod is assumed.
3. For structural problems, PTUBE entries may only reference MAT1 material entries.

PVISC - *Viscous Damper Element Properties*

Description

Defines the viscous properties of a one-dimensional viscous element which is used to create viscous elements by means of the CVISC entry.

Format

1	2	3	4	5	6	7	8	9	10
PVISC	PIDA	C1A	C2A		PIDB	C1B	C2B		

Example

PVISC	3	6.2	3.94						

Field	Description
PIDi	Property identification number. (Integer > 0)
C1i, C2i	Viscous coefficients for extension and rotation. (Real)

Remarks

1. PVISC entries must have unique identification numbers with respect to all other property entries.
2. This entry is used for both extensional and rotational viscous elements.
3. Up to two viscous damper element property sets may be defined on a single entry.
4. Viscous damper elements are applicable in direct dynamics (i.e., non-modal) problems only.
5. Viscous properties are material, and temperature, independent.

RANDPS - *Power Spectral Density Specification*

Description

Defines load set power spectral density factors for use in random analysis having the frequency dependent form:

$$S_{jk}(f) = (x + iy)G(f)$$

Format

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

Example

RANDPS	5	3	7	2.0	2.5	4			

Field	Description
SID	Random analysis set identification number. (Integer > 0)
J	Subcase identification number of excited load set. (Integer > 0)
K	Subcase identification number of applied load set. (Integer ≥ 0; K ≥ J)
X, Y	Components of complex number. (Real)
TID	Identification number of a TABRNDi entry which defines G(F). (Integer ≥ 0)

Remarks

1. If J = K, then Y must be 0.0.
2. For TID = 0, G(F) = 1.0.
3. Set identification numbers must be selected with the Case Control command RANDOM = SID.
4. Up to 20 unique sets may be defined. However, as many RANDPS entries as desired with the same SID may be input.
5. RANDPS can only reference subcases included within a single loop (change in direct matrix input is not allowed).

RANDT1 - *Autocorrelation Function Time Lag*

Description

Defines time lag constants for use in random analysis autocorrelation function computation.

Format

1	2	3	4	5	6	7	8	9	10
RANDT1	SID	N	T0	TMAX					

Example

RANDT1	5	10	3.2	9.6					

Field	Description
SID	Random analysis set identification number. (Integer > 0)
N	Number of time lag intervals. (Integer > 0)
T0	Starting time lag. (Real ≥ 0.0)
TMAX	Maximum time lag. (Real > T0)

Remarks

1. At least one RANDPS entry with the same set identification number must be present.
2. The time lags defined on this entry are given by:

$$T_i = T_o + \frac{T_{max} - T_o}{N}(i - 1), \quad i = 1, N + 1$$

3. Time lag sets must be selected with the Case Control command RANDOM = SID.

RBAR - *Rigid Bar*

Description

Defines a rigid bar with six degrees of freedom at each end.

Format

1	2	3	4	5	6	7	8	9	10
RBAR	EID	G1	G2	IC1	IC2	DC1	DC2		

Example

RBAR	5	1	2	234	123				

Field	Description
EID	Element identification number. (Integer > 0)
Gi	Identification numbers of connected grid points. (Integer > 0)
ICi	Independent degrees of freedom at grid point Gi. See Remark 2. (Integer or blank; any unique combination of the integers 1 through 6, no embedded blanks.)
DCi	Dependent degrees of freedom at grid point Gi. See Remarks 3 and 4. (Integer or blank; any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. All degrees of freedom on this entry are expressed relative to the global coordinate system.
3. The total number of degrees of freedom specified (IC1 and IC2) must equal six; for example, IC1 = 1236, IC2 = 34. Further, they should together be capable of representing any general rigid body motion of the element.
4. If both DC1 and DC2 are zero or blank, all of the degrees of freedom not in IC1 and IC2 will be made dependent.
5. The dependent (that is, constrained) degrees of freedom in an RBAR element may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set. Degrees of freedom declared to be independent by a rigid element can, however, be made dependent by another rigid element or by an MPC entry.
6. Rigid elements, unlike MPCs, are not selected through Case Control.
7. Forces of constraint are not recovered.

RBE1 - *Rigid Body Element, Form 1*

Description

Defines a rigid body connected to an arbitrary number of grid points.

Format

1	2	3	4	5	6	7	8	9	10
RBE1	EID	IG1	IC1	IG2	IC2	IG3	IC3		
		IG4	IC4	IG5	IC5	IG6	IC6		
	“UM”	DG1	DC1	DG2	DC2	DG3	DC3		
		DG4	DC4	DG5	DC5	-etc.-			

Example

RBE1	103	11	1	12	2	13	4		
		14	35	15	6				
	UM	21	123	22	1	23	123456		
		24	456	25	2				

Field	Description
EID	Element identification number. (Integer > 0)
IGi	Independent (reference) grid point identification numbers. (Integer > 0 or blank)
ICi	Independent degrees of freedom at grid points IGi. See Remarks 3, 4, and 6. (Integer or blank; any unique combination of the integers 1 through 6, no embedded blanks.)
"UM"	Character string indicating the start of the dependent grid point data.
DGi	Dependent grid point identification numbers. (Integer > 0)
DCi	Dependent degrees of freedom at grid points DGi. See Remarks 4 and 5. (Integer or blank; any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. All degrees of freedom on this entry are expressed relative to the global coordinate system.
3. The total number of degrees of freedom specified for the reference grid points (IC1 through IC6) must be six; for example, IC1=1, IC2=2, IC3=4, IC4=35, IC5=6. Further, they should together be capable of representing any general rigid body motion of the element.
4. The first continuation entry is not required if less than four independent reference grid points are specified.
5. The dependent (that is, constrained) degrees of freedom in an RBE1 element may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set. Degrees of freedom declared to be independent by a rigid element can, however, be made dependent by another rigid element or by an MPC entry.
6. A degree of freedom cannot be both independent and dependent for the same element. However, both independent and dependent components can exist at the same grid point.
7. Rigid elements, unlike MPCs, are not selected via Case Control.
8. Forces of constraint are not recovered.

RBE2 - *Rigid Body Element, Form 2*

Description

Defines a rigid body whose independent degrees of freedom are specified at a single grid point and whose dependent degrees of freedom are specified at an arbitrary number of grid points.

Format

1	2	3	4	5	6	7	8	9	10
RBE2	EID	IG	C	G1	G2	G3	G4	G5	
	G6	G7	G8	-etc.-					

Example

RBE2	9	8	12	10	12	14	15	16	
	20								

Field	Description
EID	Element identification number. (Integer > 0)
IG	Identification number of the reference, or independent grid point, to which all six independent degrees of freedom for the element are assigned. (Integer > 0)
C	Degree of freedom component number(s) for all dependent Gi grid points. See Remark 3. (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)
Gi	Dependent grid point identification number(s). (Integer > 0)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. All degrees of freedom on this entry are expressed relative to the global coordinate system.
3. The dependent (that is, constrained) degrees of freedom in an RBE2 element may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set. Degrees of freedom declared to be independent by a rigid element can, however, be made dependent by another rigid element or by an MPC entry.
4. Rigid elements, unlike MPCs, are not selected via Case Control.
5. Forces of constraint are not recovered.

RBE3 - *Rigid Body Element, Form 3*

Description

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

Format

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFG	REFC	W1	C1	G1,1	G1,2	
	G1,3	-etc.-	W2	C2	G2,1	G2,2	G2,3	-etc.-	
	W3	C3	G3,1	G3,2	G3,3	-etc.-			
	"UM"	DG1	DC1	DG2	DC2	DG3	DC3		
		DG4	DC4	-etc.-					

Example

RBE3	14		100	1234	1.0	123	1	3	
	5	4.7	1	2	4	6	5.2	2	
	7	8		5.1	1	15	16		
	UM	100	14	5	3	7	2		

Field	Description
EID	Element identification number. (Integer > 0)
REFG	Reference grid point identification number. (Integer > 0)
REFC	Reference grid degree of freedom components whose values are to be computed by the weighted average. (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)
Wi	Weighting factor for components of motion at grid points Gi,j. (Real)
Ci	Degree of freedom components to be weighted by factor Wi at grid points Gi,j. (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)
Gi,j	Grid points whose components Ci have weighting factor Wi in the averaging equations. (Integer > 0)
"UM"	Character string indicating the start of optional data for components of motion at dependent grid points DGi. See Remark 4. (Character)The default behavior is only those REFC degrees of freedom at the REFG reference grid are included in the dependent degree of freedom set (*m*-set).
DGi	Dependent grid point identification number(s) (grid points with displacement degrees of freedom in the *m*-set.) (Integer > 0).
DCi	Degree of freedom component(s) at dependent grid point DGi. (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. All degrees of freedom on this entry are expressed relative to the global coordinate system.
3. Blank spaces may be left at the end of a Gi,j sequence.

4. The default behavior is only those REFC degrees of freedom at the REFG reference grid are included in the dependent degree of freedom set (*m*-set). The “UM” flag indicates the start of a list of alternate *m*-set degrees of freedom. If the “UM” option is used:

   (a) The total number of components in the *m*-set (that is, the total number of dependent degrees of freedom defined by the element) must be equal to the number of components in REFC (four in the above example).

   (b) The components in UM must be a subset of the components listed in REFC and (Gi,j; Ci).

   (c) The coefficient matrix [Rm] in the constraints equation [Rm]{um} + [Rn]{un} = 0 must be nonsingular.

4. The dependent (that is, constrained) degrees of freedom in an RBE2 element may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set. Degrees of freedom declared to be independent by a rigid element can, however, be made dependent by another rigid element or by an MPC entry.
5. RBE3 elements, unlike MPCs, are always active and are not selected through Case Control.
6. Forces of constraint are not recovered.

RELES - *Release Substructure Connectivities*

Description

Defines substructure grid point degree of freedom sets which are not to be connected.

Format

1	2	3	4	5	6	7	8	9	10
RELES	SID	NAME	G1	C1	G2	C2	G3	C3	
	G4	C4	-etc.-						

Example

RELES	6	SWINGRT	17	456	18	456	21	123	
	253	456							

Field	Description
SID	Connectivity set identification number.
NAME	Name of basic substructure. (Character)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. RELES entries will override any connections generated automatically from geometry and any connections defined on CONCT entries.
2. The RELES entry will not override connections defined on CONCT1 entries.
3. Connectivity sets must be selected in the Substructure Control section (CONNECT = SID) to be used by NASTRAN-CORE. Note that CONNECT is a subcommand of the substructure COMBINE command.
4. Connectivities defined during previously executed COMBINE operations will be retained and may be referenced by the grid point ID and component of any one of the basic substructures associated with that connectivity.

RFORCE - *Rotational Force*

Description

Defines a static loading condition due to an angular velocity.

Format

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	N1	N2	N3		
	RACC								

Example

RFORCE	2	5		-6.4	0.0	0.0	1.0		
	2.5								

Field	Description
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. See Remark 1. (Integer ≥ 0)
CID	Coordinate system for defining rotation vector components. (Integer ≥ 0; Default = 0)
A	Scale factor for rotational velocity in revolutions per unit time. (Real)
N1, N2, N3	Rectangular components of rotation direction vector. The vector defined will act at point G. (Real; $N1^2 + N2^2 + N3^2 > 0.0$)
RACC	Angular acceleration per unit time in revolutions per unit time squared.

Remarks

1. The force applied to grid G is given by:

$$\{F\}_i = \{F_a\} + \{F_t\} = [M]_i\{\vec{\omega} \times (\vec{\omega} \times \vec{r}_i) + \vec{\alpha} \times \vec{r}_i\}$$

2. G = 0 implies the basic coordinate system origin.
3. CID = 0 indicates the basic coordinate system.
4. Load sets must be selected in Case Control (LOAD = SID) to be used in static analysis.
5. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
6. Rotational force sets can be combined with other static loads only by using the LOAD Bulk Data entry.
7. The load vector generated by this entry can be printed with an OLOAD request in Case Control.
8. For elements with lumped mass, the centrifugal acceleration is calculated at the center of the lumped mass. Grid point offsets of the mass such as those defined with CBAR and CONM2 elements are taken into account.
9. For elements using the coupled consistent mass option (COUPMASS) or those with implicit coupled mass matrices such as the TRIAAX elements, the centrifugal accelerations are calculated based on grid point locations. This acceleration vector is then multiplied by the mass matrix to generate loads. Therefore, for greater accuracy, elements near the axis of rotation should be kept small to best represent the actual acceleration field.
10. When applying a rotational force to an axisymmetric element, G and CID must be 0 or blank; N1 and N2 must be 0.0

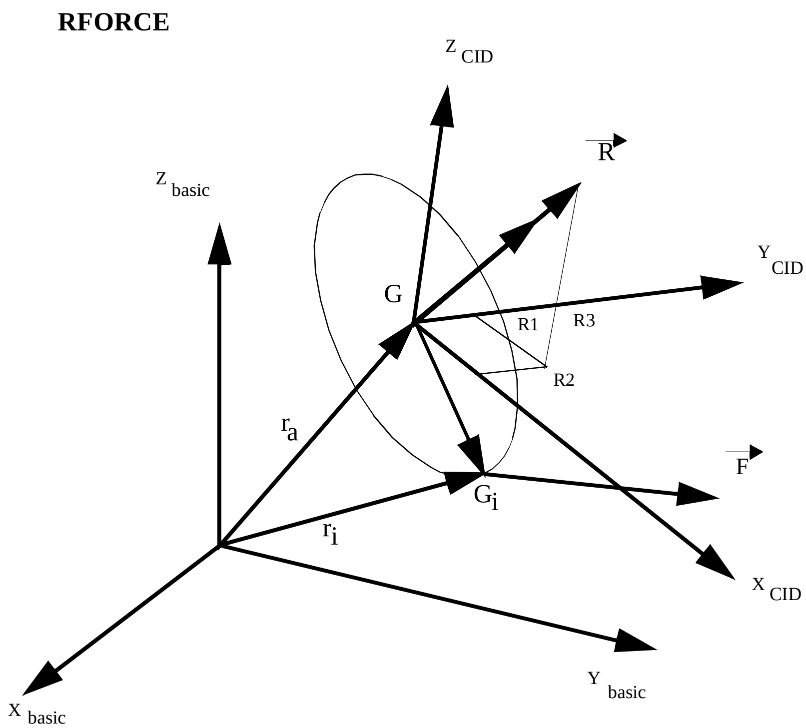


Figure 4-42 Rotational Force Resultants

RINGAX - *Axisymmetric Ring*

Description

Defines a ring for a model containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
RINGAX	ID		R	Z			PS		

Example

RINGAX	3		2.0	-10.0			162		

Field	Description
ID	Ring identification number. (Integer > 0)
R	Ring radius. (Real > 0.0)
Z	Ring axial location. (Real)
PS	Permanent single-point constraints. (Integer; any unique combination of the integers 1 through 6 with no embedded blanks)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. The number of degrees of freedom defined for the ring is (6-PS)*H where PS is the number of constraints applied in field 8 and H is the harmonic count defined on the AXIC entry.
3. RINGAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
4. The fourth and sixth degrees of freedom must be constrained when transverse shear flexibility is not included for the conical shell.
5. For a discussion of the conical shell problem see Section 5.9 of the Theoretical Manual.

6. For a discussion of the axisymmetric solid problem see Section 5.11 of the Theoretical Manual.

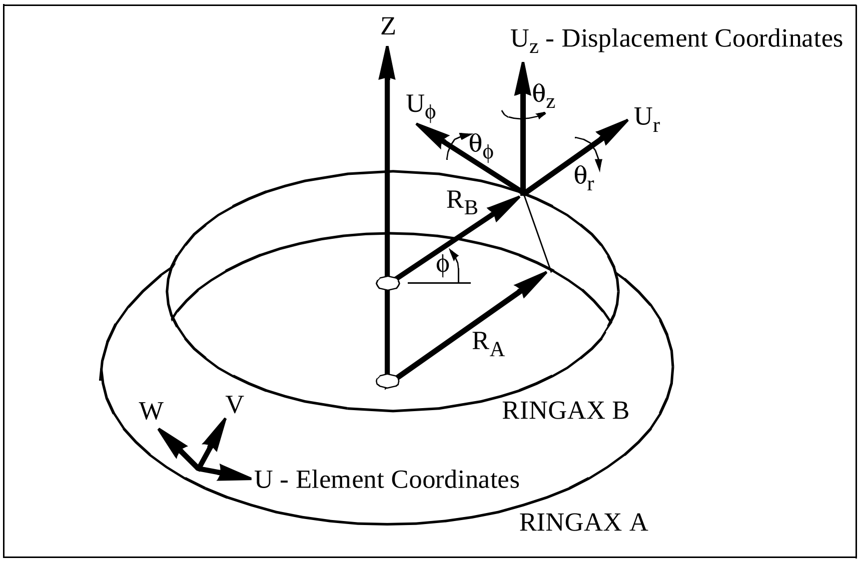


Figure 4-43 RINGAX Coordinate System

RLOAD1 - *Frequency Response Dynamic Load, Form 1*

Description

Defines a frequency dependent dynamic load of the form:

$$\{P(f)\} = \{A \cdot [C(f) + iD(f)]e^{i(\theta - 2\pi f \tau)}\}$$

for use in frequency response problems.

Format

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	LOADID	DELAY	DPHASE	TC	TD			

Example

RLOAD1	5	3	6	9	1	2			

Field	Description
SID	Set identification number. See Remark 3. (Integer > 0)
LOADID	Identification number of a static load including DEFORM, FORCE, FORCE1, FORCE2, GRAV, MOMENT, MOMENT1, MOMENT2, MOMENT3, PLOAD, PLOAD1, PLOAD2, PLOAD4, SLOAD,TEMP, TEMPD, TEMP1, TEMP2, TEMPRB, or of a DAREA entry set which defines the load coefficient, *A*. See Remarks 4 and 5. (Integer > 0)
DELAY	Identification number of a DELAY or DELAYS entry set which defines τ. See Remark 2. (Integer ≥ 0 or blank)
DPHASE	Identification number of a DPHASE or DPHASES entry set which defines θ. See Remark 2. (Integer ≥ 0 or blank)
TC	Set identification number of a TABLEDi entry which gives *C(f)*. See Remark 2. (Integer ≥ 0 or blank)
TD	Set identification number of a TABLEDi entry which gives *D(f)*. See Remark 2. (Integer ≥ 0 or blank)

Remarks

1. Dynamic load sets must be selected in Case Control (DLOAD = SID).
2. If any of DELAY, DPHASE, TC, or TD are blank or zero, the corresponding τ, θ, *C(f)*, or *D(f)* will be zero. Either TC or TD may be blank or zero but not both.
3. RLOAD1 and RLOAD2 loads may only be combined by specification on a DLOAD entry. Note that this implies that RLOAD1 and RLOAD2 entry SIDs must be unique. Further, SIDs must be unique across all RLOAD1,2 and TLOAD1,2 entries.
4. For automated multi-stage substructuring, LOADID (field 3) references a DAREAS entry set which, in turn, may only reference degrees of freedom in the boundary set of the solution structure.
5. For automated multi-stage substructuring, the LOADID field may also reference LOADC entries. In this case, DAREAS entries with the same set identification and nonzero loads must also exist.
6. Static loads generate external loads only for g-set degrees of freedom. Only DAREA entries can generate loads on extra points.

RLOAD2 - *Frequency Response Dynamic Load, Form 2*

Description

Defines a frequency dependent dynamic load of the form:

$$\{P(f)\} = \{A \cdot B(f) e^{i(\phi(f) + \theta - 2\pi f \tau)}\}$$

for use in frequency response problems.

Format

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	LOADID	DELAY	DPHASE	TB	TP			

Example

RLOAD2	5	3	6	21	7	2			

Field	Description
SID	Set identification number. See Remark 3. (Integer > 0)
LOADID	Identification number of a static load including DEFORM, FORCE, FORCE1, FORCE2, GRAV, MOMENT, MOMENT1, MOMENT2, MOMENT3, PLOAD, PLOAD1, PLOAD2, PLOAD4, SLOAD,TEMP, TEMPD, TEMP1, TEMP2, TEMPRB, or of a DAREA entry set which defines the load coefficient, *A*. See Remarks 4 and 5. (Integer > 0)
DELAY	Identification number of a DELAY or DELAYS entry set which defines τ. See Remark 2. (Integer ≥ 0 or blank)
DPHASE	Identification number of a DPHASE or DPHASES entry set which defines θ in degrees. See Remark 2. (Integer ≥ 0 or blank)
TB	Set identification number of a TABLEDi entry which gives $B(f)$. (Integer > 0)
TP	Set identification number of a TABLEDi entry which gives $\phi(f)$ in degrees. See Remark 2. (Integer ≥ 0 or blank)

Remarks

1. Dynamic load sets must be selected in Case Control (DLOAD = SID).
2. If any of DELAY, DPHASE, or TP are blank or zero, the corresponding τ, θ, or $\phi(f)$ will be zero.
3. RLOAD1 and RLOAD2 loads may only be combined by specification on a DLOAD entry. Note that this implies that RLOAD1 and RLOAD2 entry SIDs must be unique. Further, SIDs must be unique across all RLOAD1,2 and TLOAD1,2 entries.
4. For automated multi-stage substructuring, LOADID (field 3) references a DAREAS entry set which, in turn, may only reference degrees of freedom in the boundary set of the solution structure.
5. For automated multi-stage substructuring, the LOADID field may also reference LOADC entries. In this case, DAREAS entries with the same set identification and nonzero loads must also exist.
6. Static loads generate external loads only for g-set degrees of freedom. Only DAREA entries can generate loads on extra points.

RROD - *Rigid Rod Element Connection*

Description

Defines a pin-ended rod element that is rigid in extension-compression.

Format

1	2	3	4	5	6	7	8	9	10
RROD	EIDA	GA	G1A	C1A	EIDB	GB	G1B	C1B	

Example

RROD	104	5	9	3	302	12	4	2	

Field	Description
EIDi	Element identification number. (Integer > 0)
Gi	Reference grid point identification number. (Integer > 0)
G1i	Dependent grid point identification number. (Integer > 0; G1 ≠ G)
C1i	Dependent translational degree of freedom of grid point G1. (1 ≤ Integer ≤ 3)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Only one reference grid point and only one dependent grid point are allowed per element. The two points may not be coincident.
3. The direction represented by the dependent translational degree of freedom of the dependent grid point may not be perpendicular or nearly perpendicular to the element.
4. Up to two RROD elements may be defined on a single entry.
5. The dependent degrees of freedom in an RROD element may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set. Degrees of freedom declared to be independent by a rigid element can, however, be made dependent by another rigid element or by an MPC entry..
6. For a discussion of rigid elements, see Section 3.5.6 of the Theoretical Manual.

RSPLINE - *Interpolation Constraint Element*

Description

Defines multipoint constraints for the interpolation of displacements at a grid point.

points.

Format

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
	C4	G5	C5	G6	-etc.-				

Example

RSPLINE	73	.05	27	28	123456	29		30	
	123	75	123	71					

Field	Description
EID	Element identification number. (Integer > 0)
D/L	Ratio of the diameter of the elastic tube which the spline represents to the sum of the lengths of all segments (Real > 0.0; Default = 0.1).
Gi	Identification number of the ith grid point. (Integer > 0)
Ci	Component(s) to be constrained at Gi. See Remark 4. (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Displacements are interpolated from the equations of an elastic beam passing through the grid points.
3. All degrees of freedom on this entry are expressed relative to the global coordinate system.
4. A blank entry in Ci indicates that all six degrees of freedom at Gi are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last entry must be a Gi, not a Ci. For the example shown, G1, G3, and G6 are independent; G2 has six constrained degrees of freedom while G4 and G5 each have three.
5. The dependent degrees of freedom in an RSPLINE element may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set. Degrees of freedom declared to be independent by a rigid element can, however, be made dependent by another rigid element or by an MPC entry.
6. Rigid elements, unlike MPCs, are not selected through Case Control.
7. Forces of constraint are not recovered.
8. The RSPLINE is not really a rigid element in the normal sense, and should not be used for other than its intended purpose.

RTRPLT - *Rigid Triangular Plate*

Description

Defines a rigid triangular plate element.

Format

1	2	3	4	5	6	7	8	9	10
RTRPLT	EID	G1	G2	G3	IC1	IC2	IC3		
	DC1	DC2	DC3						

Example

RTRPLT	7	1	2	3	1236	3	3		

Field	Description
EID	Element identification number. (Integer > 0)
Gi	Identification numbers of the connected grid points. (Integer > 0)
ICi	Independent degrees of freedom at grid points Gi. See Remark 3. (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)
DCi	Dependent degrees of freedom at grid points Gi. See Remarks 4 and 5. (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. Each element identification number must be unique with respect to all other element identification numbers.
2. All degrees of freedom on this entry are expressed relative to the global coordinate system.
3. The total number of degrees of freedom specified for the reference grid points (IC1, IC2, and IC3) must be six; for example, IC1 = 1236, IC2 = 3, IC3 = 3. Further, they should together be capable of representing any general rigid body motion of the element.
4. If DC1, DC2, and DC3 are all zero or blank or if the continuation entry is omitted, all of the degrees of freedom not in IC1, IC2, and IC3 will be made dependent.
5. The dependent degrees of freedom in an RTRPLT element may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set. Degrees of freedom declared to be independent by a rigid element can, however, be made dependent by another rigid element or by an MPC entry.
6. Rigid elements, unlike MPCs, are not selected through Case Control.
7. Forces of constraint are not recovered.

SECTAX - *Axisymmetric Sector*

Description

Defines a sector of a model containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
SECTAX	ID	RID	R	PHI1	PHI2				

Example

1	2	3	4	5	6	7	8	9	10
SECTAX	1	2	3.0	30.0	40.0				

Field	Description
ID	Sector identification number. (Integer > 0)
RID	Identification number of a RINGAX entry. (Integer > 0)
R	Effective radius. (Real)
PHIi	Azimuthal limits of sector in degrees. (Real)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. SECTAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.

SLOAD - *Static Scalar Load*

Description

Defines static loads on scalar points.

Format

1	2	3	4	5	6	7	8	9	10
SLOAD	SID	S1	F1	S2	F2	S3	F3		

Example

SLOAD	16	2	5.9	17	-6.3	14	-2.93		

Field	Description
SID	Load set identification number. (Integer > 0)
Si	Scalar point identification numbers. (Integer > 0)
Fi	Load value. (Real)

Remarks

1. Load sets must be selected using the Case Control command LOAD = SID in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. Up to three scalar loads may be defined on a single entry.

SPC - *Single-Point Constraints*

Description

Defines sets of single-point constraints and enforced displacements.

Format

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

Example

SPC	2	32	436	-2.6	5		+2.9		

Field	Description
SID	Single-point constraint set identification number. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Di	Value of enforced displacement for all components designated by Gi and Ci pair. (Real)

Remarks

1. The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
2. Di must be 0.0 for dynamics problems.
3. Single-point forces of constraint are recovered during stress data recovery.
4. Single-point constraint sets must be selected in Case Control (SPC = SID) to be used by NASTRAN-CORE.
5. Up to twelve single-point constraints may be defined on a single entry.
6. SPC degrees of freedom may be redundantly specified as permanent constraints in the PS field of a GRID entry.
7. The enforced displacement, D, is used only in static analyses (Rigid Formats 1, 2, 4, 5, 6, 14).

SPC1 - *Single-Point Constraints, Alternate Form*

Description

Defines sets of single-point constraints.

Format

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example

SPC1	3	2	1	3	10	9	6	5	
	2	8							

Alternate Format and Example

SPC1	SID	C	G1	“THRU”	G2				
SPC1	313	12456	6	THRU	32				

Field	Description
SID	Single-point constraint set identification number. (Integer > 0)
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Gi	Grid or scalar point identification numbers. (Integer > 0)

Remarks

1. Unlike the SPC entry, note that enforced displacements are not available on this entry. As many continuation entries as necessary may be used when THRU is not used.
2. The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
3. Single-point constraint sets must be selected in Case Control (SPC = SID) to be used by NASTRAN-CORE.
4. SPC degrees of freedom may be redundantly specified as permanent constraints in the PS field of a GRID entry.
5. When using the “THRU” option, all grid points in the range of G1 through G2 must exist.

SPCADD - *Single-Point Constraint Set Combination*

Description

Defines a single-point constraint set as a union of single-point constraint sets defined via SPC and/or SPC1 entries.

Format

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

Example

SPCADD	100	3	2	9	1				

Field	Description
SID	New single-point constraint set identification number. (Integer > 0, SID ≠ 101 or 102 if axisymmetric.)
Si	Identification numbers of single-point constraint sets defined via SPC or SPC1 entries. (Integer > 0; SID ≠ Si)

Remarks

1. Single-point constraint sets must be selected in Case Control (SPC = SID) to be used by NASTRAN-CORE.
2. Single-point constraint set identification numbers must be unique with respect to all other single-point constraint set identification numbers.
3. Set identification numbers of 101 or 102 cannot be used in axisymmetric problems.
4. The SPCADD entry does not operate on SPCD entries, as these are loading entries.

SPCAX - *Axisymmetric Single-Point Constraint*

Description

Defines sets of single-point constraints for axisymmetric models containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
SPCAX	SID	RID	HID	C	V				

Example

SPCAX	2	3	4	13	6.0				

Field	Description
SID	Single-point constraint set identification number. Integer > 0; SID ≠ 101 or 102)
RID	RINGAX entry identification number (see RINGAX). (Integer ≥ 0)
HID	Harmonic identification number. (Integer ≥ 0)
C	Component number(s). (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)
V	Value of enforced displacement. (Real)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Single-point constraint sets must be selected in Case Control (SPC = SID) to be used by NASTRAN-CORE.
3. Coordinates appearing on SPCAX entries may not appear on MPCAX, SUPAX, or OMITAX entries.

SPCD - *Enforced Displacements*

Description

Defines enforced displacements for static analysis.

Format

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

Example

SPCD	100	32	436	-2.6	5		+2.9		

Field	Description
SID	Static load set identification number. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Di	Value of enforced displacement for all components designated by Gi and Ci pair. (Real)

Remarks

1. This entry only applies in static analysis, where the load set identification number (SID) is selected using the Case Control command LOAD=SID.
2. A coordinate referenced on this entry must also be referenced by a SPC or SPC1 entry, selected in Case Control with the SPC command.
3. If the SID load set is requested in Case Control, values of Di will override the values specified on an SPC Bulk Data entry, if present.
4. Note that the LOAD Bulk Data entry cannot be used to combine SPCD's with other load entries. In addition, since static enforced displacements cannot be the only type of loading entry, at least one additional Bulk Data loading entry (FORCE, SLOAD, etc.) in addition to the SPCD entries is required in the LOAD set selected in Case Control. If the enforced displacement is the only loading in the requested LOAD set, the additional entry may be a FORCE entry defining a loading of 0.0 magnitude.

SPCS - *Substructure Single Point Constraints*

Description

Defines a single point constraint set for a basic substructure.

Format

1	2	3	4	5	6	7	8	9	10
SPCS	SID	NAME	G1	C1	G2	C2	G3	C3	
	G4	C4	G5	C5	-etc.-				

Example

SPCS	61	MIDWG	9	45	18	124	36	456	
	88	136							

Field	Description
SID	Set identification number. (Integer > 0)
NAME	Basic substructure name. (Character)
Gi	Substructure grid or scalar point identification numbers. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. Single-point constraint sets must be selected in Case Control (SPC = SID) to be used by NASTRAN-CORE.
2. The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
3. All of the referenced grid points must exist.
4. Single-point forces of constraint are recovered during stress data recovery.
5. A single G, C pair may not specify all component degrees of freedom for a connected grid point where only some of the degrees of freedom of the grid point have been connected or when some have been disconnected via the RELES entry. The degrees of freedom which were connected and those that were not connected must be referenced separately.

SPCS1 - *Substructure Single Point Constraints, Alternate Form*

Description

Defines a single point constraint set for a basic substructure.

Format

1	2	3	4	5	6	7	8	9	10
SPCS1	SID	NAME	C	G1	G2	G3	G4	G5	
	G6	G7	G8	-etc.-					

Example

SPCS1	15	FUSELAG	1236	1101	1102	1105	THRU	1110	
	1121	1130	THRU	1140	1143	1150			

Field	Description
SID	Set identification number. (Integer > 0)
NAME	Basic substructure name. (Character)
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Gi	Grid or scalar point identification numbers. (Integer > 0)

Remarks

1. Single-point constraint sets must be selected in Case Control (SPC = SID) to be used by NASTRAN-CORE.
2. The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
3. All of the referenced grid points must exist.
4. THRU may appear in fields 6, 7, or 8 of the first entry and anywhere in continuation entry fields 3 through 8.
5. Single-point forces of constraint are recovered during stress data recovery.
6. A single G, C pair may not specify all component degrees of freedom for a connected grid point where only some of the degrees of freedom of the grid point have been connected or when some have been disconnected via the RELES entry. The degrees of freedom which were connected and those that were not connected must be referenced separately.

SPCSD - *Substructure Enforced Displacements*

Description

Defines substructure enforced displacements for static analysis.

Format

1	2	3	4	5	6	7	8	9	10
SPCSD	SID	NAME	G1	C1	D1	G2	C2	D2	

Example

SPCSD	27	LWINGRT	965	3	3.6				

Field	Description
SID	Static load set identification number. (Integer > 0)
NAME	Basic substructure name. (Character)
Gi	Grid or scalar point identification numbers. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Di	Value of enforced displacement for all components designated by Gi and Ci pair. (Real)

Remarks

1. This entry only applies in static analysis, where the load set identification number (SID) is selected using the Case Control command LOAD=SID.
2. A coordinate referenced on this entry must also be referenced by a SPC or SPC1 entry, selected in Case Control with the SPC command.
3. Note that the LOAD Bulk Data entry cannot be used to combine SPCSD's with other load entries.
4. At least one Bulk Data loading entry (LOADC or SLOAD) in addition to the SPCSD must exist in the loading set (LOAD) requested in Case Control.

SPOINT - *Scalar Point*

Description

Defines scalar points (points that have only a single degree of freedom.)

Format

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

Example

SPOINT	3	18	1	4	16	2			

Alternate Format and Example

SPOINT	ID1	"THRU"	ID2						
SPOINT	5	THRU	649						

Field	Description
IDi	Scalar point identification numbers. (Integer > 0; ID1 < ID2 for "THRU" option)

Remarks

1. Scalar points implicitly defined on a scalar connection entry (e.g, CDAMPi, CELASi, etc.) need not be listed on an SPOINT entry.
2. Scalar point identification numbers must be unique with respect to all other structural and scalar points.
3. This entry is used primarily to define scalar points appearing in single or multipoint constraint equations but to which no scalar elements are connected.
4. The alternate format defines a range of scalar points ID1 through ID2, inclusive.
5. For a discussion of scalar points, see Section 5.6 of the Theoretical Manual.

SUPAX - *Axisymmetric Fictitious Support*

Description

Defines coordinates at which determinate reactions are to be applied during the analysis of a free body modeled with CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
SUPAX	RID1	HID1	C1	RID2	HID2	C2			

Example

SUPAX				4	3	2			

Field	Description
RIDi	Ring identification number. (Integer > 0)
HIDi	Harmonic identification number. (Integer ≥ 0)
Ci	Component number(s). (Integer; any unique combination of the integers 1 through 6, no embedded blanks.)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Up to 12 coordinates may appear on a single entry.
3. The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
4. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
5. For a discussion of the axisymmetric solid problem, see Section 5.11 of the Theoretical Manual.

SUPORT - *Fictitious Support*

Description

Defines coordinates at which determinate reactions are to be applied to a free body during analysis.

Format

1	2	3	4	5	6	7	8	9	10
SUPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example

SUPORT	16	215							

Field	Description
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. The degrees of freedom listed on this entry may not appear on any other entry that specifies members of a mutually-exclusive degree of freedom set.
2. Up to twenty-four support coordinates may be defined on a single entry.
3. Note that SUPORT is spelled with only a single “P”.

TABDMP1 - *Modal Damping Table*

Description

Defines modal damping as a tabular function of frequency.

Format

1	2	3	4	5	6	7	8	9	10
TABDMP1	TID	TYPE							
	F1	G1	F2	G2	F3	G3	-etc.-		

Example

TABDMP1	3								
	2.5	.01057	2.6	.01362	ENDT				

Field	Description
TID	Table identification number. (Integer > 0)
TYPE	Type of damping units. (Character: “G”, “CRIT”, or “Q; Defaults “G”)
Fi	Frequency value in cycles per unit time. (Real ≥ 0.0)
Gi	Damping value. (Real)
“ENDT”	Flag indicating end of table entries. (Character)

Remarks

1. Modal damping tables must be selected in Case Control (SDAMP = TID) to be used by NASTRAN-CORE.
2. The Fi must be in either ascending or descending order but not both.
3. At least two (Fi, Gi) points must be defined.
4. Any (Fi, Gi) table entry may be ignored by placing the string “SKIP” in either of two fields used for that entry.
5. The end of the table is indicated by an “ENDT” string in either of the two fields following the last entry.
6. TABDMP1 uses the algorithm:

$$G = g_T\langle F\rangle$$

   where F is input to the table and G is returned. The table look-up $g_T(F)$ is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (F(i+1) = Fi, with G(i+1) ≠ Gi) are allowed, except at the end points. At such discontinuities, or jump points, the average of G(i+1) and Gi is used. There are no error returns from this table look-up procedure.
7. Structural damping is used only in modal formulations of complex eigenvalue analysis, frequency response analysis, or transient response analysis.

TABLED1 - *Dynamic Load Tabular Function, Linear Form 1*

Description

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads.

Format

1	2	3	4	5	6	7	8	9	10
TABLED1	TID								
	x1	y1	x2	y2	x3	y3	x4	y4	
	-etc.-		“ENDT”						

Example

TABLED1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

Field	Description
TID	Table identification number. (Integer > 0)
xi, yi	Tabular entries. (Real)
“ENDT”	Flag indicating end of table entries. (Character)

Remarks

1. The xi must be in either ascending or descending order but not both.
2. At least two (xi, yi) points must be defined.
3. Any (xi, yi) table entry may be ignored by placing the character string “SKIP” in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of the character string “ENDT” in either of the two fields following the last entry.
5. Each TABLEDi mnemonic implies the use of a specific algorithm. For TABLED1 type tables, this algorithm is:

$$y = y_T(x)$$

   where x is input to the table and y is returned. Table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (x(i+1) = xi, with y(i+1) ≠ yi) are allowed, except at the end points. At such discontinuities, or jump points, the average of y(i+1) and yi is used. There are no error returns from this table look-up procedure.
6. Linear extrapolation is not used for Fourier Transform methods. The function is zero outside the range.

TABLED2 - *Dynamic Load Tabular Function, Linear Form 2*

Description

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1							
	x1	y1	x2	y2	x3	y3	x4	y4	
	-etc.-		"ENDT"						

Example

TABLED2	15	-10.5							
	1.0	-4.5	2.0	-4.2	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

Field	Description
TID	Table identification number. (Integer > 0)
X1	Table parameter. (Real)
xi, yi	Tabular entries. (Real)
"ENDT"	Flag indicating end of table entries. (Character)

Remarks

1. The xi must be in either ascending or descending order but not both.
2. At least two (xi, yi) points must be defined.
3. Any (xi, yi) table entry may be ignored by placing the character string "SKIP" in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry.
5. Each TABLEDi mnemonic implies the use of a specific algorithm. For TABLED2 type tables, this algorithm is

$$y = y_T(x - X1)$$

   where x is input to the table and y is returned. Table look-up $y_T(\bar{x})$ with $\bar{x} = (x - X1)$ is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (x(i+1) = xi, with y(i+1) ≠ yi) are allowed, except at the end points. At such discontinuities, or jump points, the average of y(i+1) and yi is used. There are no error returns from this table look-up procedure.
6. Linear extrapolation is not used for Fourier Transform methods. The function is zero outside the range.

TABLED3 - *Dynamic Load Tabular Function, Liner Proportional Form*

Description

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format

1	2	3	4	5	6	7	8	9	10
TABLED3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	x4	y4	
	-etc.-		“ENDT”						

Example

TABLED3	62	126.9	30.0						
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

Field	Description
TID	Table identification number. (Integer > 0)
X1, X2	Table parameters. (Real; X2 ≠ 0.0)
xi, yi	Tabular entries. (Real)
“ENDT”	Flag indicating end of table entries. (Character)

Remarks

1. The xi must be in either ascending or descending order but not both.
2. At least two (xi, yi) points must be defined.
3. Any (xi, yi) table entry may be ignored by placing the character string “SKIP” in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of the character string “ENDT” in either of the two fields following the last entry.
5. Each TABLEDi mnemonic implies the use of a specific algorithm. For TABLED3 type tables, this algorithm is:

$$y = y_T\left(\frac{x - X1}{X2}\right)$$

   where x is input to the table and y is returned. Table look-up $y_T(\bar{x})$ with $\bar{x} = \frac{(x - X1)}{X2}$, is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (x(i+1) = xi, with y(i+1) ≠ yi) are allowed, except at the end points. At such discontinuities, or jump points, the average of y(i+1) and yi is used. There are no error returns from this table look-up procedure.

6. Linear extrapolation is not used for Fourier Transform methods. The function is zero outside the range.

TABLED4 - *Dynamic Load Tabular Function, Power Series Form*

Description

Defines coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format

1	2	3	4	5	6	7	8	9	10
TABLED4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	-etc.-	"ENDT"		

Example

TABLED4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

Field	Description
TID	Table identification number. (Integer > 0)
Xi	Table parameters. (Real; X2 ≠ 0.0; X3 < X4)
Ai	Coefficient entries. (Real)
"ENDT"	Flag indicating end of table entries. (Character)

Remarks

1. At least one continuation entry must be present.
2. The end of the table is indicated by the existence of the character string "ENDT" in the field following the last entry.
3. Each TABLEDi mnemonic implies the use of a specific algorithm. For TABLED4 type tables, this algorithm is

$$y = \sum_{i=0}^{N} A_i \cdot \left(\frac{x - X1}{X2}\right)^i$$

where x is input to the table and y is returned. Whenever x < X3, use X3; whenever x > X4, use X4. There are N + 1 entries in the table. There are no error returns from this table look-up procedure.

TABLEM1 - *Material Property Table, Linear Form 1*

Description

Defines a tabular function for use in generating temperature dependent material properties.

Format

1	2	3	4	5	6	7	8	9	10
TABLEM1	TID								
	x1	y1	x2	y2	x3	y3	x4	y4	
	-etc.-		"ENDT"						

Example

TABLEM1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

Field	Description
TID	Table identification number. (Integer > 0)
xi, yi	Tabular entries. (Real)
"ENDT"	Flag indicating end of table entries. (Character)

Remarks

1. The xi must be in either ascending or descending order but not both.
2. At least two (xi, yi) points must be defined.
3. Any (xi, yi) table entry may be ignored by placing the character string "SKIP" in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry.
5. Each TABLEMi mnemonic implies the use of a specific algorithm. For TABLEM1 type tables, this algorithm is

$$y = y_T(x)$$

where *x* is input to the table and *y* is returned. Table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (x(i+1) = xi, with y(i+1) ≠ yi) are allowed, except at the end points. At such discontinuities, or jump points, the average of y(i+1) and yi is used. There are no error returns from this table look-up procedure.

TABLEM2 - *Material Property Table, Linear Form 2*

Description

Defines a tabular function for use in generating temperature dependent material properties. Also contains parametric data for use with the table.

Format

1	2	3	4	5	6	7	8	9	10
TABLEM2	TID	X1							
	x1	y1	x2	y2	x3	y3	x4	y4	
	-etc.-		“ENDT”						

Example

TABLEM2	15	-10.5							
	1.0	-4.5	2.0	-4.5	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

Field	Description
TID	Table identification number. (Integer > 0)
X1	Table parameter. (Real)
xi, yi	Tabular entries. (Real)
“ENDT”	Flag indicating end of table entries. (Character)

Remarks

1. The xi must be in either ascending or descending order but not both.
2. At least two (xi, yi) points must be defined.
3. Any (xi, yi) table entry may be ignored by placing the character string “SKIP” in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of the character string “ENDT” in either of the two fields following the last entry.
5. Each TABLEMi mnemonic implies the use of a specific algorithm. For TABLEM2 type tables, this algorithm is:

$$y = Z \cdot y_T(x - X1)$$

where x is input to the table, y is returned, and Z is supplied from the basic MATi entry. Table look-up $y_T(\bar{x})$ with $\bar{x} = (x - X1)$ is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (x(i+1) = xi, with y(i+1) ≠ yi) are allowed, except at the end points. At such discontinuities, or jump points, the average of y(i+1) and yi is used. There are no error returns from this table look-up procedure.

TABLEM3 - *Material Property Table, Linear Proportional Form*

Description

Defines a tabular function for use in generating temperature dependent material properties. Also contains parametric data for use with the table.

Format

1	2	3	4	5	6	7	8	9	10
TABLEM3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	x4	y4	
	-etc.-		"ENDT"						

Example

TABLEM3	62	126.9	30.0						
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

Field	Description
TID	Table identification number. (Integer > 0)
X1, X2	Table parameters. (Real; X2 ≠ 0.0)
xi, yi	Tabular entries. (Real)
"ENDT"	Flag indicating end of table entries. (Character)

Remarks

1. The xi must be in either ascending or descending order but not both.
2. At least two (xi, yi) points must be defined.
3. Any (xi, yi) table entry may be ignored by placing the character string "SKIP" in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry.
5. Each TABLEMi mnemonic implies the use of a specific algorithm. For TABLEM3 type tables, this algorithm is

$$y = Z \cdot y_T\left(\frac{x - X1}{X2}\right)$$

where x is input to the table, y is returned, and Z is supplied from the basic MATi entry. Table look-up $y_T(\bar{x})$ with $\bar{x} = \frac{(x - X1)}{X2}$, is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (x(i+1) = xi, with y(i+1) ≠ yi) are allowed, except at the end points. At such discontinuities, or jump points, the average of y(i+1) and yi is used. There are no error returns from this table look-up procedure.

TABLEM4 - *Material Property Table, Power Series Form*

Description

Defines coefficients of a power series for use in generating temperature dependent material properties. Also contains parametric data for use with the table.

Format

1	2	3	4	5	6	7	8	9	10
TABLEM4	ID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	-etc.-	"ENDT"		

Example

TABLEM4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

Field	Description
ID	Table identification number. (Integer > 0)
Xi	Table parameters. (Real; X2 ≠ 0.0; X3 < X4)
Ai	Coefficient entries. (Real)
"ENDT"	Flag indicating end of table entries. (Character)

Remarks

1. At least one continuation entry must be present.
2. The end of the table is indicated by the existence of the character string "ENDT" in the field following the last entry.
3. Each TABLEMi mnemonic implies the use of a specific algorithm. For TABLEM4 type tables, this algorithm is:

$$y = \sum_{i=0}^{N} A_i \cdot \left(\frac{x - X1}{X2}\right)^i$$

where x is input to the table, y is returned, and Z is supplied from the basic MATi entry. Whenever x < X3, use X3; whenever x > X4, use X4. There are N + 1 entries in the table. There are no error returns from this table look-up procedure.

TABLES1 - *Tabular Stress-Strain Function*

Description

Defines a tabular stress-strain function for use in piecewise linear analysis.

Format

1	2	3	4	5	6	7	8	9	10
TABLES1	TID								
	x1	y1	x2	y2	x3	y3	x4	y4	
	-etc.-		"ENDT"						

Example

TABLES1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

Field	Description
TID	Table identification number. (Integer > 0)
xi, yi	Tabular entries. (Real)
"ENDT"	Flag indicating end of table entries. (Character)

Remarks

1. The xi must be in either ascending or descending order but not both.
2. For piecewise linear analysis, the yi numbers must form a non-decreasing sequence for an ascending xi sequence and vice versa.
3. At least two (xi, yi) points must be defined.
4. Any (xi, yi) table entry may be ignored by placing the character string "SKIP" in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry.
6. The TABLES1 interpolation algorithm is simply,

$$y = y_T(x)$$

   where x is input to the table and y is returned. Table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (x(i+1) = xi, with y(i+1) ≠ yi) are allowed, except at the end points. At such discontinuities, or jump points, the average of y(i+1) and yi is used. There are no error returns from this table look-up procedure.

7. The tabular stress-strain function may only have a zero slope at its end.

TABRND1 - *Power Spectral Density Table*

Description

Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced on the RANDPS entry.

Format

1	2	3	4	5	6	7	8	9	10
TABRND1	TID								
	f1	g1	f2	g2	f3	g3	f4	g4	
	-etc.-		“ENDT”						

Example

TABRND1	3								
	2.5	.01057	2.6	.01362	ENDT				

Field	Description
ID	Table identification number. (Integer > 0)
fi	Frequency value in cycles per unit time. (Real ≠ 0.0)
gi	Power spectral density. (Real)
“ENDT”	Flag indicating end of table entries. (Character)

Remarks

1. The fi must be in either ascending or descending order but not both.
2. At least two (fi, gi) points must be defined.
3. Any (fi, gi) table entry may be ignored by placing the string “SKIP” in either of two fields used for that entry.
4. The end of the table is indicated by an “ENDT” string in either of the two fields following the last entry.
5. The TABRND1 interpolation algorithm is simply,

$$g = g_T(f)$$

   where *f* is input to the table and *g* is returned. Table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. Table discontinuities (f(i+1) = fi, with g(i+1) ≠ gi) are allowed, except at the end points. At such discontinuities, or jump points, the average of g(i+1) and gi is used. There are no error returns from this table look-up procedure.

TEMP - *Grid Point Temperature Field*

Description

Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress data recovery.

Format

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G1	T1	G2	T2	G3	T3		

Example

TEMP	3	94	316.2	49	219.8				

Field	Description
SID	Temperature set identification number. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ti	Grid point temperature. (Real)

Remarks

1. Temperature sets must be selected in Case Control (TEMP = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. Up to three grid point temperatures may be defined on a single entry.
4. The temperature set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in Case Control.
5. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP2, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
6. If the element material is temperature dependent, its properties are evaluated at the average temperature. In the case of isoparametric hexahedron elements, their properties are evaluated at the temperature computed by interpolating the grid point temperatures.
7. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined.

TEMPAX - *Axisymmetric Temperature*

Description

Defines temperature sets for a model containing CCONEAX, CTRAPAX, or CTRIAAX elements.

Format

1	2	3	4	5	6	7	8	9	10
TEMPAX	SID1	RID1	PHI1	TEMP1	SID2	RID2	PHI2	TEMP2	

Example

TEMPAX	4	7	30.0	105.3					

Field	Description
SIDi	Temperature set identification number. (Integer >0)
RIDi	Identification number of a RINGAX entry. (Integer > 0)
PHIi	Azimuthal angle in degrees. (Real)
TEMPi	Temperature. (Real)

Remarks

1. This entry is allowed only if an AXIC entry is also present.
2. Up to two temperatures may be defined on a single entry.
3. Temperature sets must be selected in Case Control (TEMP = SID) to be used by NASTRAN-CORE.
4. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in Case Control.
5. At least two different angles are required for each RID and temperature set to specify the subtended angle [$\phi_b - \phi_a$] over which the temperature applies.
6. For a discussion of the conical shell problem, see Section 5.9 of the Theoretical Manual.
7. For a discussion of the axisymmetric solid problem, see Section 5.11 of the Theoretical Manual.

TEMPD - *Grid Point Temperature Field Default*

Description

Defines a temperature default for all grid points of the structural model which have not been assigned a temperature on a TEMP entry.

Format

1	2	3	4	5	6	7	8	9	10
TEMPD	SID1	T1	SID2	T2	SID3	T3	SID4	T4	

Example

TEMPD	1	216.3							

Field	Description
SIDi	Temperature set identification number. (Integer > 0)
Ti	Default temperature. (Real)

Remarks

1. Temperature sets must be selected in Case Control (TEMP = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. Up to four default temperatures may be defined on a single entry.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP2, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. If the element material is temperature dependent its properties are evaluated at the average temperature. In the case of isoparametric hexahedron elements, their properties are evaluated at the temperature computed by interpolating the grid point temperatures.
6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined.
7. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in Case Control.

TEMPP1 - *Plate Element Temperature Field*

Description

Defines a temperature field for plate, membrane, and combination elements (by an average temperature and a thermal gradient over the cross-section) for determination of thermal loading, temperature-dependent material properties, or stress data recovery

Format

1	2	3	4	5	6	7	8	9	10
TEMPP1	SID	EID1	T	TPRIME	T1	T2			
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example

TEMPP1	2	24	62.0	10.0	57.0	67.0			
	26	21	19	30					

Continuation Entry Alternate Format and Example

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
	1	THRU	10	30	THRU	61			

Field	Description
SID	Temperature set identification number. (Integer > 0)
EIDn	Unique element identification number(s). The continuation entry may have “THRU” in fields 3 and/or 6, in which case EID2 < EIDi, EIDj < EIDk. (Integer > 0 or character “THRU”)
T	Average temperature over the cross-section. Assumed constant over area. (Real)
TPRIME	Effective linear thermal gradient. Not used for membranes. (Real)
T1, T2	Temperatures for stress calculation, at points defined on the element property entry. These data are not used for membrane elements. (Real)

Remarks

1. Temperature sets must be selected in Case Control (TEMP = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. Elements specified on the continuation entries are used in addition to EID1. Elements must not be specified more than once.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP2, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. For a temperature field other than a constant gradient the effective gradient for a homogeneous plate is:

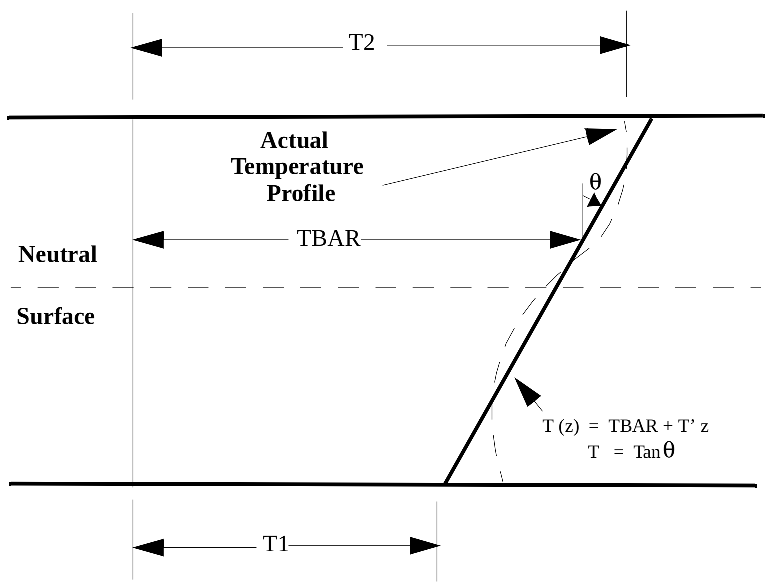
$$T' = \frac{1}{1}\int_z T(z)z\ dz$$

where I is the bending inertia, and z is the distance from the neutral surface in the positive normal direction.

6. The average temperature for a homogeneous plate is

$$T = \frac{1}{Volume} \int_{Volume} T \, dVolume$$

7. If the element material is temperature dependent, its properties are evaluated at the average temperature T.
8. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in Case Control.



TEMPP2 - *Plate Element Temperature Field*

Description

Defines a temperature field for plate, membrane, and combination elements by an average temperature and thermal moments for determination of thermal loading, temperature-dependent material properties, or stress data recovery.

Format

1	2	3	4	5	6	7	8	9	10
TEMPP2	SID	EID1	T	MX	MY	MXY	T1	T2	
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example

TEMPP2	2	36	68.8						
	400	1	2	5					

Continuation Entry Alternate Format and Example

	EID2	"THRU"	EIDi	EIDj	"THRU"	EIDk			
	37	THRU	312	315	THRU	320			

Field	Description
SID	Temperature set identification number.
EIDn	Unique element identification number(s). A continuation entry may have "THRU" in field 3 and/or 6 in which case EID2 < EIDi, EIDj < EIDk. (Integer > 0 or character "THRU")
T	Average temperature over cross-section. Assumed constant over area. (Real)
MX, MY, MXY	Resultant thermal moments per unit width in element coordinate system. Not used for membrane elements. (Real)
T1, T2	Temperature for stress calculation at points defined on the element property entry. These data are not used for membrane elements. (Real)

Remarks

1. Temperature sets must be selected in Case Control (TEMP = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. Elements specified on the continuation entries are used in addition to EID1. Elements must not be specified more than once.
4. If thermal effects are requested all elements must have a temperature field defined either directly on a TEMPP1, TEMPP2, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. The thermal moments in the element coordinate system may be calculated from the formula:

$$\begin{Bmatrix} \{M_x\} \\ \{M_y\} \\ \{M_{xy}\} \end{Bmatrix} = -\int [G_e]\{\alpha_e\}T(z)zdz$$

where the integration is performed over the bending material properties in the element coordinate system.

$[G_e]$	3x3 Elastic coefficient matrix
$\{\alpha_e\}$	3x1 Material thermal expansion coefficients
$T(z)$	Temperature at z
z	Distance from the neutral surface in the element coordinate system.

6. The temperature dependent material properties are evaluated at the average temperature T. If a property varies with depth, an effective value must be used which satisfies the desired elastic and stress relationships. The temperatures at the fiber distances may be changed to compensate for local differences in $\{\alpha_e\}$ and produce correct stresses.
7. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in Case Control.

TEMPP3 - *Plate Element Temperature Field*

Description

Defines a temperature field for homogeneous plate, membrane, and combination elements (by a tabular description of the thermal field over the cross-section) for determination of thermal loading, temperature-dependent material properties, or stress data recovery.

Format

1	2	3	4	5	6	7	8	9	10
TEMPP3	SID	EID1	Z0	T0	Z1	T1	Z2	T2	
	Z3	T3	Z4	T4	Z5	T5	Z6	T6	
	Z7	T7	Z8	T8	Z9	T9	Z10	T10	
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example

TEMPP3	17	39	0.0	32.9	2.0	43.4	2.5	45.0	
	3.0	60.0	4.0	90.0					
	1	2	3	4	5	6	8	10	

Third and Subsequent Continuation Entry Alternate Format and Example

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
				1	THRU	10			

Field	Description
SID	Temperature set identification number. (Integer > 0)
EIDn	Unique element identification number(s). The continuation entry may have “THRU” in fields 3 and/or 6 in which case EID2 < EIDi, EIDj < EIDk. (Integer > 0 or character “THRU”)
Z0	Position of the bottom surface with respect to an arbitrary reference plane. (Real)
Zi	Positions on cross-section from bottom to top of cross-section relative to the arbitrary reference plane. There must be an increasing sequence with the last nonzero value corresponding to the top surface. (Real)
T0	Temperature at the bottom surface. (Real)
Ti	Temperature at position Zi. (Real)

Remarks

1. Temperature sets must be selected in Case Control (TEMP = SID) to be used in static analysis.
2. In dynamic analysis the set of static loads associated with this entry is generated automatically if the entry is specified in Bulk Data. The associated static load generated for the entry is then used in dynamic analysis if the set indentification number of the entry is referenced by the LOADID field of a RLOAD1, RLOAD2, TLOAD1 or TLOAD2 dynamic load entry and that entry is in turn referenced by a DLOAD Case Control directive.
3. Elements specified on the third and succeeding continuation entries are used in addition to EID1. Elements must not be specified more than once.
4. The first and second continuation entries must be present if a list of elements is to be used.

5. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP2, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
6. If the element material is temperature dependent, its properties are evaluated at the average temperature over the depth which is calculated by the program using a linear distribution between points.
7. The data is limited to a maximum of eleven points on the temperature-depth profile.
8. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in Case Control.

TEMPRB - *One-Dimensional Element Temperature Field*

Description

Defines a temperature field for the CBAR, CBEAM, CELBOW, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress data recovery.

Format

1	2	3	4	5	6	7	8	9	10
TEMPRB	SID	EID1	TA	TB	TP1a	TP1b	TP2a	TP2b	
	TCa	TDa	TEa	TFa	TCb	TDb	TEb	TFb	
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example

TEMPRB	200	1	68.0	23.0	0.0	28.0		2.5	
	68.0	91.0	45.0		48.0	80.0	20.0		
	9	10							

Second and Subsequent Continuation Entry Alternate Format and Example

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
	2	THRU	4	10	THRU	14			

Field	Description
SID	Temperature set identification number. (Integer > 0)
EIDn	Unique element identification number(s). The second continuation entry may have “THRU” in fields 3 and/or 6 in which case EID2 < EID1, EIDj < EIDk. (Integer > 0 or character “THRU”)
TA, TB	Average temperature over the area at end a and end b. (Real)
TPij	Effective linear gradient in direction i on end j (BAR only.) (Real)
Tij	Temperatures at point i as defined on the PBAR entries at end j. These data are used for stress recovery only (BAR only). (Real)

Remarks

1. Temperature sets must be selected in Case Control (TEMP = SID) to be used by NASTRAN-CORE.
2. If at least one nonzero or nonblank Tij is present, the point temperatures given are used for stress recovery. If no Tij values are given, linear temperature gradients are assumed for stresses.
3. Elements specified on the third and succeeding continuation entries are used in addition to EID1. Elements must not be specified more than once.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP2, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. The effective thermal gradients in the element coordinate system for the BAR element are defined by the following integrals over the cross-section. For end a (end b is similar):

$$T'_{1a} = \frac{1}{I_1}\int_A T_a(y,z)y\,dA$$

$$T'_{2a} = \frac{1}{I_2}\int_A T_a(y,z)z\ dA$$

where $T_a(y,z)$ is the temperature at point y,z (in the element coordinate system) at end a of the BAR. I_1 and I_2 are the moment of inertia about the z and y axis, respectively. The temperatures are assumed to vary linearly along the length (x-axis). Note that if the temperature varies linearly over the cross-section then T'_{1a}, T'_{1b}, T'_{2a}, and T'_{2b} are the actual gradients.

6. If the element material is temperature dependent, the material properties are evaluated at the average temperature

$$\frac{TA + TB}{2}$$

except for the BEAM element, which assumes the temperature varies linearly from end A to end B and the material properties are evaluated at the interpolated termperature at the internal points specified on the PBEAM.

7. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in Case Control.

TF - *Dynamic Transfer Function*

Description

Defines a transfer function of the form:

$$(B0 + B1p + B2p^2)u_d + \sum_i (A0(i) + A1(i)p + A2(i)p^2)u_i = 0$$

May also be used as a means of direct matrix input.

Format

1	2	3	4	5	6	7	8	9	10
TF	SID	GD	CD	B0	B1	B2			
	G(1)	C(1)	A0(1)	A1(1)	A2(1)				
	-etc.-								

Example

TF	1	2	3	4.0	5.0	6.0			
	3	4	5.0	6.0	7.0				

Field	Description
SID	Set identification number. (Integer > 0)
GD, G(i)	Grid, scalar, or extra point identification number. (Integer > 0)
CD, C(i)	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
B0, B1, B2; A0(i), A1(i), A2(i)	Transfer function coefficients. (Real)

Remarks

1. The matrix elements defined by this entry are added to the dynamic matrices for the problem.
2. Transfer function sets must be selected in Case Control (TFL = SID) to be used by NASTRAN-CORE.
3. Only one entry (one SID) is allowed for each GD, CD combination.
4. The constraint relation given above will hold only if no elements are connected to the dependent coordinate.

TIC - *Transient Initial Condition*

Description

Defines values for the initial conditions of coordinates used in transient analysis. Both displacement and velocity values may be specified at independent coordinates of the structural model.

Format

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0				

Example

TIC	1	3	2	5.0	-6.0				

Field	Description
SID	Set identification number. (Integer > 0)
G	Grid, scalar or extra point identification number. (Integer > 0)
C	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
U0	Initial displacement. (Real)
V0	Initial velocity. (Real)

Remarks

1. Transient initial condition sets must be selected with the Case Control command IC = SID.
2. If a TIC set is not selected in Case Control, all initial conditions are assumed zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed zero.
4. Initial conditions may be used only in direct transient analysis and may only be applied to the analysis of degrees of freedom, that is, only those coordinates that are not constrained on other entries (e.g., MPC, SPC, OMIT.)

TICS - *Transient Initial Condition, Substructure Analysis*

Description

Defines values for the initial conditions of coordinates used in direct transient analysis. Both displacement and velocity values may be specified at independent coordinates of the structural model.

Format

1	2	3	4	5	6	7	8	9	10
TICS	SID	NAME	G	C	U0	V0			

Example

TICS	1	SPAR	3	2	5.0	-6.0			

Field	Description
SID	Set identification number. (Integer > 0)
NAME	Basic substructure name. (Character)
G	Grid, scalar or extra point identification number. (Integer > 0)
C	Component number. (Integer; zero or blank for scalar points, any one of the integers 1 through 6 for grid points, no embedded blanks.)
U0	Initial displacement value. (Real)
V0	Initial velocity value. (Real)

Remarks

1. Transient initial condition sets must be selected in Case Control (IC = SID) to be used by NASTRAN-CORE.
2. If a TIC set is not selected in Case Control, all initial conditions are assumed zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed zero.
4. Initial conditions may only be used direct transient analysis (Rigid Format 9) and may only be applied to the analysis of degrees of freedom, that is, only those coordinates retained in the solution substructure and not constrained using MPC, SPC, or OMIT entries.
5. This entry is used in the substructure SOLVE operation.

TLOAD1 - *Transient Response Dynamic Load, Form 1*

Description

Defines a time-dependent dynamic load of the form

$$\{P(t)\} = \{A \cdot F(t - \tau)\}$$

for use in transient response problems.

Format

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	LOADID	DELAY		TF				

Example

TLOAD1	5	7	9		13				

Field	Description
SID	Set identification number. See Remark 4. (Integer > 0)
LOADID	Identification number of a static load including DEFORM, FORCE, FORCE1, FORCE2, GRAV, MOMENT, MOMENT1, MOMENT2, MOMENT3, PLOAD, PLOAD1, PLOAD2, PLOAD4, SLOAD,TEMP, TEMPD, TEMP1, TEMP2, TEMPRB, or of a DAREA entry set which defines the load coefficient, *A*. See Remarks 4 and 5. (Integer > 0)
DELAY	Identification number of DELAY or DELAYS entry set which defines τ. (Integer $\geq$ 0)
TF	Identification number of a TABLEDi entry which gives $F(t - \tau)$. (Integer > 0)

Remarks

1. If DELAY is zero, τ will be zero.
2. Field 5 must be blank.
3. Dynamic load sets must be selected in Case Control (DLOAD = SID).
4. TLOAD2 and TLOAD1 loads may only be combined by specification on a DLOAD entry. Note that this implies that TLOAD1 and TLOAD2 entry SIDs must be unique. Further, SIDs must be unique across all TLOAD1,2 and RLOAD1,2 entries.
5. For automated multi-stage substructuring, LOADID (field 3) references a DAREAS entry set which, in turn, may only reference degrees of freedom in the boundary set of the solution structure.
6. For automated multi-stage substructuring, the LOADID field may also reference LOADC entries. In this case, DAREAS entries with the same set identification and nonzero loads must also exist.
7. Static loads generate external loads only for g-set degrees of freedom. Only DAREA entries can generate loads on extra points.

TLOAD2 - *Transient Response Dynamic Load, Form 2*

Description

Defines a time-dependent dynamic load of the form:

$$\{P(t)\} = \begin{cases} \{0\} & ; \quad t < (T1 + \tau),\ t > (T2 + \tau) \\ \{A \cdot \tilde{t}^B e^{C\tilde{t}} \cos(2\pi F\tilde{t} + \phi)\} & ; \quad (T1 + \tau) \le t \le (T2 + \tau) \end{cases}$$

for use in transient response problems, where $\tilde{t} = t - T1 - \tau$

Format

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	LOADID	DELAY		T1	T2	F	PHI	
	C	B							

Example

TLOAD2	4	10	7		2.1	4.7	12.0	30.0	
	2.0	3.0							

Field	Description
SID	Set identification number. See Remark 4. (Integer > 0)
LOADID	Identification number of a static load including DEFORM, FORCE, FORCE1, FORCE2, GRAV, MOMENT, MOMENT1, MOMENT2, MOMENT3, PLOAD, PLOAD1, PLOAD2, PLOAD4, SLOAD,TEMP, TEMPD, TEMP1, TEMP2, TEMPRB, or of a DAREA entry set which defines the load coefficient, *A*. See Remarks 4 and 5. (Integer > 0)
DELAY	Identification number of a DELAY or DELAYS entry set which defines τ. (Integer ≥ 0)
T1	Time constant. (Real ≥ 0.0)
T2	Time constant. (Real, T2 > T1)
F	Frequency in cycles per unit time. (Real ≥ 0.0; Default = 0.0)
PHI	Phase angle in degrees. (Real; Default = 0.0)
C	Exponential coefficient. (Real; Default = 0.0)
B	Growth coefficient. (Real; Default = 0.0)

Remarks

1. If DELAY is zero, τ will be zero.
2. Field 5 must be blank.
3. Dynamic load sets must be selected in Case Control (DLOAD = SID).
4. TLOAD2 and TLOAD1 loads may only be combined by specification on a DLOAD entry. Note that this implies that TLOAD1 and TLOAD2 entry SIDs must be unique. Further, SIDs must be unique across all TLOAD1,2 and RLOAD1,2 entries.
5. For automated multi-stage substructuring, LOADID (field 3) references a DAREAS entry set which, in turn, may only reference degrees of freedom in the boundary set of the solution structure.

6. For automated multi-stage substructuring, the LOADID field may also reference LOADC entries. In this case, DAREAS entries with the same set identification and nonzero loads must also exist.
7. Static loads generate external loads only for g-set degrees of freedom. Only DAREA entries can generate loads on extra points.

TRANS - *Component Substructure Transformation Definition*

Description

Defines the location and orientation of the component substructure basic coordinate system relative to the combined substructure.

Format

1	2	3	4	5	6	7	8	9	10
TRANS	SID		A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example

TRANS	1		0.0	0.0	0.0	0.0	-0.5	10.0	
	0.0	10.0	0.5						

Field	Description
SID	Set identification number. (Integer > 0)
Ai, Bi, Ci	Coordinates of points defining coordinate system orientation. See Remark 1. (Real; points A, B, and C must be unique and non-colinear)

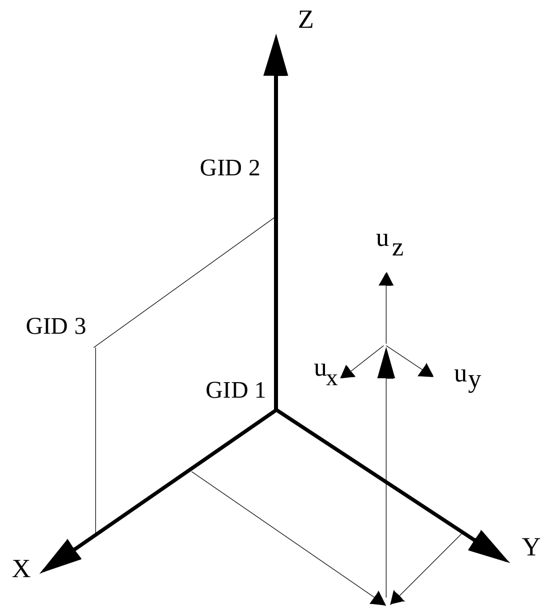
Remarks

1. The location and orientation of the component substructure basic coordinate system is defined relative to the basic coordinate system of the COMBINEd substructure. The coordinates of points A, B, and C, expressed in terms of the combined substructure basic coordinate system, are defined as follows:

Ai	Defines the location of the origin of the basic coordinate system of the component substructure.
Bi	Defines the location of a point on the z axis of the basic coordinate system of the component substructure.
Ci	Defines the location of a point in the positive xz plane of the basic coordinate system of the component substructure.

2. The continuation entry must be present.
3. SID must be unique with respect to all other TRANS entries.
4. Transformation sets for a whole substructure must be selected in Substructure Control (TRANS = SID) to be used by NASTRAN-CORE. Note that TRANS is a subcommand of the substructure COMBINE command.

5. Component substructure individual grid point displacement transformations are requested by the GTRAN Bulk Data entry which references TRANS information.



TSTEP - *Transient Analysis Time Steps*

Description

Defines time step intervals for analysis and data recovery in a transient analysis.

Format

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N1	DT1	NO1					
		N2	DT2	NO2					
		-etc.-							

Example

TSTEP	2	10	.001	5					
		9	0.01	1					

Field	Description
SID	Set identification number. (Integer > 0)
Ni	Number of time steps of value DTi. (Integer ≥ 1)
DTi	Time increment. (Real > 0.0)
NOi	Output skip factor; every NOi-th step will be saved for output. (Integer > 0)

Remarks

1. TSTEP entries must be selected in Case Control (TSTEP = SID) in order to be used by NASTRAN-CORE.

USERDATA - *User-defined bulk data entry*

Description

Provides an entry for use in user-provided DMAP modules.

Format

1	2	3	4	5	6	7	8	9	10
USERDATA	user input	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-	
	-etc.-	-etc.-							

Example

USERDATA	A	B	3	5.9	DATA	TED	14	18	
	32.999								

Field	Description
user input	any user-defined data using standard NASTRAN-xMG format

Remarks

1. This entry is provided to allow user-defined input for user-provided modules.
2. There is no format checking on the input in this entry.
3. As many USERDATA entries as desired may be used. The entries are written into the GEOM1 table using a Record Header of 1201,12,389.
4. Each entry is copied directly into the record in GEOM1. The record length is open-ended (with a limit of 1000 fields).
5. Each entry will be ended in GEOM1 with a series of three -1 terms. As such, it is recommended not to have three consecutive fields with a value of -1 in them, as this may be mis-interpreted in the user-provided routines.

USET - *User-defined dof set membership*

Description

Defines degrees of freedom to be placed in a user-designated set.

Format

1	2	3	4	5	6	7	8	9	10
USET	SETNAME	ID1	C1	ID2	C2	ID3	C3		

Example

USET	U1	1	1234	5	35				

Field	Description
SETNAME	Set name - (BCD - the name of one of the sets used internally in the program)
IDi	GRID or SPOINT id (integer >0)
Ci	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)

Remarks

1. Degrees of freedom specified USET entries are placed in the requested sets with no checking performed to verify consistency. If you choose to place dof in conflicting sets (for example, placing a dof in both the M- and N- sets) the program may fail without warning.
2. This entry is processed by the USETPRT module and the results appear in any set-definition tables printed using the USETPRT parameter.
3. Note - if DIAG 21 or 22 are used, the set definition tables are printed before the application of AUTOSPC or USET/USET1 entries..
4. There are 3 user-defined sets (U1, U2, and U3) available, which are not used in standard processing.

USET1 - *User-defined dof set membership*

Description

Defines degrees of freedom to be placed in a user-designated set.

Format

1	2	3	4	5	6	7	8	9	10
USET1	SETNAME	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

Example

USET1	A	123	1	5	7	8	14	18	
	21								

Alternate Format and Example

USET1	SETNAME	C	G1	"THRU"	G2				
USET1	U1	0	17	THRU	109				

Field	Description
SETNAME	Set name - (BCD - the name of one of the sets used internally in the program)
C	Component number(s). (Integer; zero or blank for scalar points, any unique combination of the integers 1 through 6 for grid points, no embedded blanks.)
Gi	Grid or scalar point identification numbers. (Integer > 0; G1 < G2 when using the "THRU" option)

Remarks

1. Degrees of freedom specified USET entries are placed in the requested sets with no checking performed to verify consistency. If you choose to place dof in conflicting sets (for example, placing a dof in both the M- and N- sets) the program may fail without warning.
2. This entry is processed by the USETPRT module and the results appear in any set-definition tables printed using the USETPRT parameter.
3. Note - if DIAG 21 or 22 are used, the set definition tables are printed before the application of AUTOSPC or USET/USET1 entries.
4. There are 3 user-defined sets (U1, U2, and U3) available, which are not used in standard processing.

5 STRUCTURAL PLOTTING

Introduction

Structural Plotter commands are used to define plots of the finite element model and its GRID point based structural responses. The actual Structural Plots are not created during your NASTRAN-CORE execution. Rather, the plot information is placed on a file that is postprocessed.

NASTRAN-CORE generates two plot files, .PLT1 and .PS, when plots are requested. The PLT1 file, am ascii plot file, can be interpreted by a user-modifiable “plt2ps” program included in the NASTRAN-CORE deliverable. The PS file is readily printable by any Postscript printer. If you do not have a Postscript printer, there are several methods available to convert Postscript file to pdf files. Some web sites which discuss this or have products are:

1. http://www.adobe.com/
2. http://www.ps2pdf.com/
3. http://www.cs.wisc.edu/~ghost/

In some cases, you mignt have to contact your NASTRAN-CORE Support Specialist for details describing how your site interfaces with this postprocessor.

This section describes the Structural Plotter terminology, the input data requirements for creating such plots, and provides examples which illustrate the use of many of the plotter commands.

Structural Plotting Terminology

This section reviews important plotting concepts that will assist you in creating plots of your model and its solution results.

Plotter Coordinate Systems. When performing graphics, there is an underlying coordinate system which is called the plotter coordinate system. The actual structural model coordinate system is mapped to the plotter system prior to plotting. The plotter coordinate system is called the *RST-System*. This system is fixed with respect to the plot you are generally looking down the R axis toward the plotter system origin. Initially, the model coordinate system is aligned with the plotter coordinate system as shown in **Figure 5-1**. Rotations are always performed with respect to the RST-System. The angles of rotation are shown in the figure.

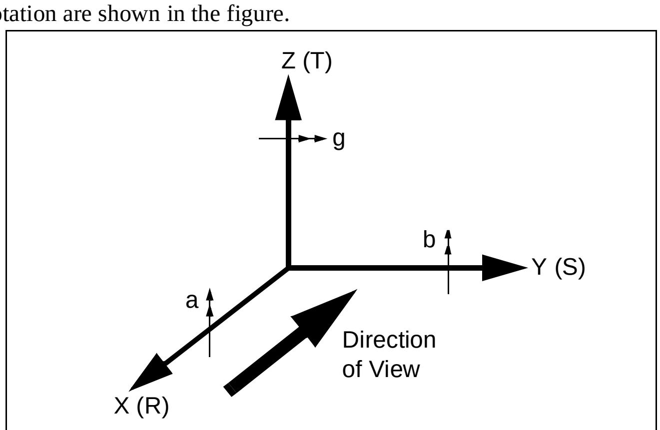


Figure 5-1 Plotter Coordinate System

Graphic Projections. When three-dimensional objects are plotted, there are numerous ways in which their coordinates may be mapped to a two-dimensional plotting surface. NASTRAN-CORE supports two of these, the *orthographic projection* and the *perspective projection.* When an object is projected to the surface along parallel lines, it is called orthographic. If, on the other hand, the object is projected along lines that converge to a point, it is called perspective. These two cases are illustrated in **Figure 5-2**.

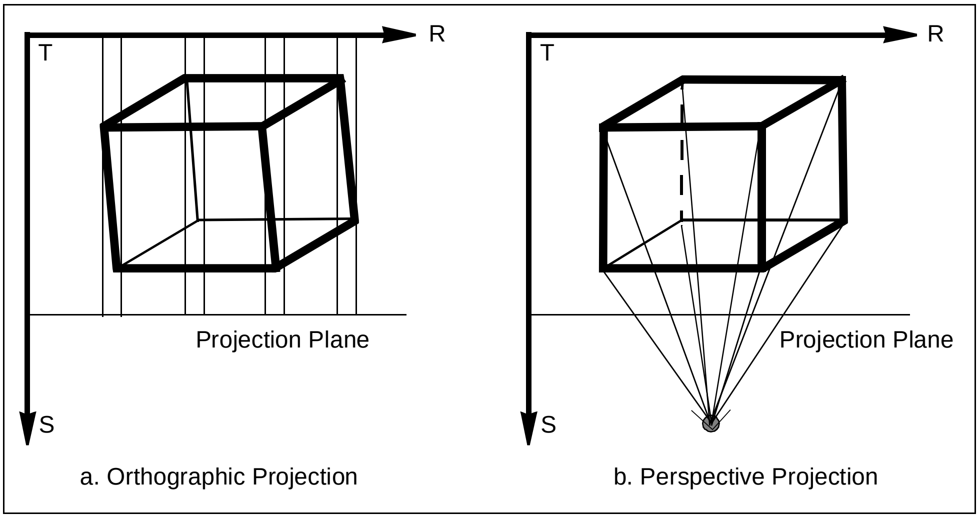


Figure 5-2 Graphic Projections

Vantage Point. As seen in **Figure 5-2b**, the point at which the lines of a perspective projection converge is called the *Vantage Point*. The vantage point is used for two purposes. For both the orthographic and perspective projections, it allows you to define the RST-coordinates of your vantage point. This allows you to view your model from a number of different angles. This feature provides an alternative to defining the angles of rotation of the model. When you use the perspective projection, the vantage point also controls the amount of perspective that you see. The closer you are to your model, the more perspective distortion in the plot.

Plot Sets. A plot set defines a group of elements within your model. Plot sets are specified by using the SET command.

Plot. A plot is a single frame which displays one or more *subplots* and, optionally, their solution results. Each of the subplots is defined as a separate plot set and each may have their own graphical characteristics.

Deformed Plot. A deformed plot is one which is created by applying the static deformations to the GRID point locations prior to plotting. Because the deformations are often very small relative to the model, they are scaled in order to be visible. When performing dynamic response analyses, you may also represent the velocities and accelerations as deformations.

Vector Plot. Rather than creating a deformed plot, you may also request that the deformations be plotted in the form of vectors. The vectors originate at the GRID points and point in a direction which is determined by the displacement components that you select.

Labels. You may selectively label the GRID points and finite elements within your model. This may be done when plotting the structural model or its deformed shape.

Coordinate System Triad. Each plot includes a coordinate system triad which shows the orientation of your model coordinate system with respect to the plot surface.

Structural Plotter Data Requirements

This section provides you with a description of the NASTRAN-CORE input data required for creating Structural Plots.

Executive Control Commands

There are no specific Executive Control commands which are required to perform Structural Plotting.

Substructure Control Commands

When you use the Substructuring capability, the BASIC command has an optional subcommand called SAVEPLOT which defines a set of elements which may be used for creating plots during a PHASE 2 operation. When using this subcommand, your PHASE 1 data stream must include a Structural Plotter subpacket which includes the definition of the referenced set.

Case Control Commands

The Structural Plotter commands form a subpacket within the Case Control command packet as shown in **Figure 5-3**.

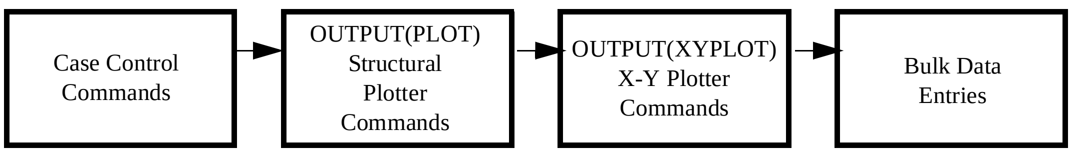


Figure 5-3 The Structural Plotter Subpacket

Both the Structural Plotter subpacket and the X-Y Plotter subpacket may be interchanged in the data stream. It is not necessary for you to issue Case Control output request commands when you are requesting DEFORMED or VECTOR Plots. NASTRAN-CORE automatically extracts the necessary data from the solution results.

Bulk Data

There is a single Bulk Data entry related to Structural Plotting. The PLOTEL entry, which is not an actual modeling element, allows you to define straight lines which connect two GRID points. You may then select these lines when creating your plots. This feature is useful for providing customized capabilities not explicitly supported by the Structural Plotter, such as the outline of your structural model.

Structural Plot Elements

Figure 5-4 shows a typical structural plot. The plot is annotated to indicate the various plot elements. Some of these are controlled by commands that you may use, while others are automatically generated by NASTRAN-CORE. The next section describes these commands in more detail.

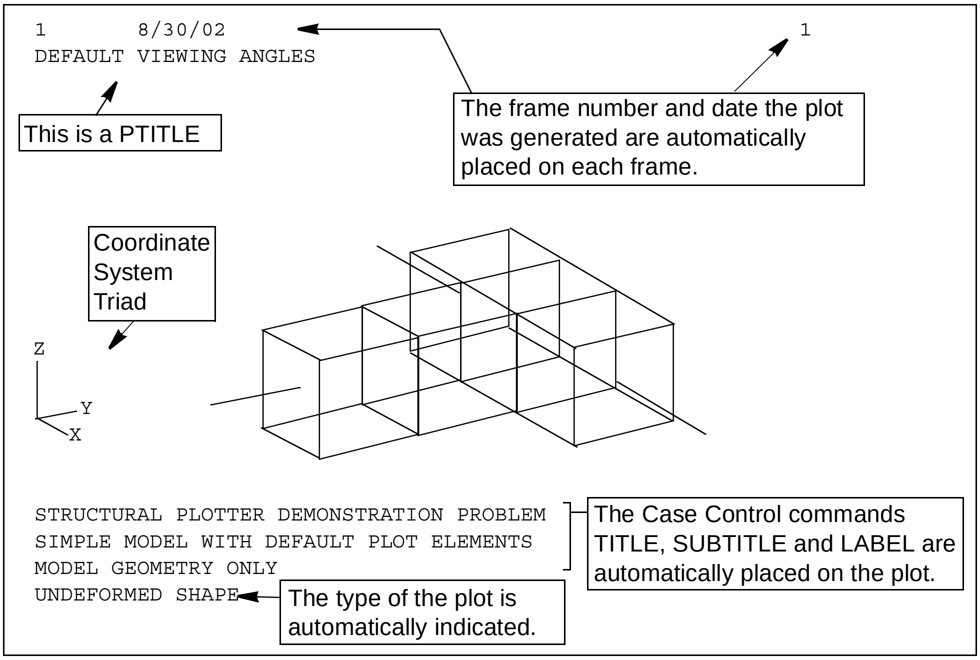


Figure 5-4 Basic Structural Plot Elements

Structural Plotter Commands

The general form of the Structural Plotter Command packet is:

```
OUTPUT(PLOT)
   option_command
   option_command
   ...
   PLOT_command
      option_command
      option_command
PLOT_command

OUTPUT(XYPLOT) or BEGIN BULK
```

The package must begin with the OUTPUT(PLOT) command and it ends when either an OUTPUT(XYPLOT) or BEGIN BULK command is encountered. The other commands in the packet define the plot frames that you with to create. Some of these are option_commands which define the characteristics of the plots to be made while others are the PLOT_commands which actually specify what is to be plotted.

Note: When performing Multidisciplinary Design Optimization, only a single Structural Plotter Command packet is used. The requested plots are generated for each appropriate discipline at each design iteration.

The available Structural Plotter Commands are summarized in **Table 5-1** where they are grouped in functional categories. The remainder of this section will provide you with an overview of these commands.

Table 5-1 Structural Plotter Commands

Commands Which:	Command Name
Define Plot Sets	SET
Control Viewing Options	AXES PERSPECTIVE FACTOR PROJECTION VANTAGE POINT VIEW ZOOM
Solution Scaling	DEFORMATION SCALE
Create Plots	PLOT subplot_list
DEFINE TITLES	PTITLE

Set Definition

You may plot all of your model or selected portions of the model which are defined using the SET command. The Structural Plotter SET command is similar to the Case Control command, but considerably more flexible. The general form of the command is:

```
SET set_id include_part [modifier_part]
```

The set_id is a unique integer identification number which is used to reference the set in subsequent Structural Plotter commands. The include_part defines the collection of elements that will be members of the set. Its general form is:

$$\left\{\begin{matrix} \text{ALL} \\ \\ \{\text{INCLUDE}\} \left\{\begin{matrix}\text{element\_type}\\ \text{element\_id}\\ \text{element\_range}\end{matrix}\right\}, \left[\left\{\begin{matrix}\text{element\_type}\\ \text{element\_id}\\ \text{element\_range}\end{matrix}\right\}\ldots\right] \end{matrix}\right\}$$

You may include ALL elements in the model, or you may select one or more element_types.

You may also select elements by their individual identification numbers, element_id, or by specifying an element_range of the form:

```
element_id_1 THRU element_id_2
```

The modifier_part may then be used to modify the include_part by either adding new element types or identifiers, or by excluding selected elements from the previous include_part. The syntax of the modifier_part is:

$$\left\{\begin{matrix}\text{EXCLUDE}\\ \text{EXCEPT}\\ \text{INCLUDE}\end{matrix}\right\} \left\{\begin{matrix}\text{element\_type}\\ \text{element\_id}\\ \text{element\_range}\end{matrix}\right\}, \left[\left\{\begin{matrix}\text{element\_type}\\ \text{element\_id}\\ \text{element\_range}\end{matrix}\right\}\ldots\right]$$

Note that the EXCLUDE and EXCEPT options are synonymous and may be used interchangeably. You may select all of the elements in your model by defining:

```
SET n = ALL
```

However, as you will see later, this is not necessary because the Structural Plotter default is to plot the entire model.

Viewing Option Commands

The viewing option commands are used to specify the characteristics, or elements, of the plot that you wish to create. The three most important viewing options are described in this section.

Viewing Angles. Figure 5-4 shows a typical structural plot with its plot elements annotated. The basic viewing angles, which is shown in the figure, are:

$$(\gamma, \beta, \alpha) = (34.27°, 23.17°, 0.0°)$$

Remember that the order in which the rotations are performed: first γ, the rotation about T, then β, the rotation about S, and finally α, the rotation about R, is crucial. The rotations are specified by the VIEW command. The special command AXES allows you to perform 90^o rotations quickly by simply specifying the correspondence between your model's coordinate system and the plotter system.

The viewing angles may also be changed by moving your position relative to the plotter coordinate system. This is done by using the VANTAGE POINT command. It is important that you remember the order in which these three commands are performed. The model coordinate system is first aligned with the plotter coordinate system with the AXES command. The VIEW command is then used to perform specified rotations of the model. Finally, your location is moved to the specified VANTAGE POINT.

The Graphics Projections. The Structural Plotter provides you with two graphics projections. You select the type of projection by specifying either:

```
ORTHOGRAPHIC PROJECTION or
PERSPECTIVE PROJECTION
```

When you request PERSPECTIVE PROJECTION plots, the amount of perspective distortion is determined by your distance from the model. This distance may be determined in one of three ways. Firstly, if you use the defaults, NASTRAN-CORE automatically determines this distance. Secondly, you may use the PERSPECTIVE FACTOR command to directly specify the amount of perspective distortion that you want. This is used to automatically compute the distance to the model. Finally, you may use the VANTAGE POINT command to move your location in space which determines the distance to the model. Note that if you wish to move your vantage point while maintaining the same perspective distortion, you should use both commands.

ZOOMing. The final viewing option command is ZOOM. This command allows you to enlarge or reduce the size of the structure about any point that you select.

Solution Scaling

When you create deformed plots, you may control the scale factor which is applied to the actual displacements by using the DEFORMATION SCALE command. You use this command to allows small deformations to become visible and to allow the deformed shape to be distinguished from the undeformed shape.

Defining a Plot Title

The command PTITLE allows you to define an additional lot title element, as shown in **Figure 5-4**.

Plotting the Model Geometry

Once you have defined the general viewing options, or allowed NASTRAN-CORE to select defaults, and defined any SETs that you wish to use, you may create one or more plots of your model. This is done with the command:

```
PLOT [global_options] [subplot_list]
```

The command allows you to specify global_options, which control the plot line_style, select labeling and symbol options, and to define the subplots within your model that are to be plotted in the same frame. The subplot_list allows you to define any number of subsets of your model and define different characteristics for each of them.

You may use one or more of the SETs you have defined in your PLOT command. The syntax of the subplot_list is:

```
subplot_list Í subplot_term, subplot_term, ...
```

Each subplot_term in the subplot_list selects a SET and defines plotting options for it. The general form of the subplot_term is:

```
subplot_term Í SET setid [set_options]
```

The set_options are similar to the global_options, but they only apply to the previously appearing SET.

Plotting Solution Results

You may also plot geometry-based solution results using the Structural Plotter. Such plots are called deformed shapes of the structural model. In this case, deformations include the grid point displacements and, for dynamic response analyses, the velocities and accelerations. The deformed plots may be made for selected subcases, time ranges, or frequency ranges, depending on the solution discipline that you are using. The general syntax of the PLOT command, when used for solution results, is:

```
PLOT results_type [UNDEFORMED] [subcase_list] [solution_range] [plot_type]
  [global_options] [subplot_list]
```

This form of the plot command is quite different from that used to plot the structural model. First, you must select the results_type to be plotted. This may be displacements, velocities, or accelerations depending on the solution discipline that you are using. You may choose to plot the UNDEFORMED shape as well as the deformed shape. Then, again depending on the analysis discipline, you may plot the solution results for selected subcases or a solution_range of times or frequencies. You also select a plot_type. The two types are DEFORMED, which draws the deformed structural model, and VECTOR, which represents the deformation as a vector. In both cases, the UNDEFORMED shape has also been plotted. The subplot_list differs from that used for model plotting in that each SET may specify a different plot_type.

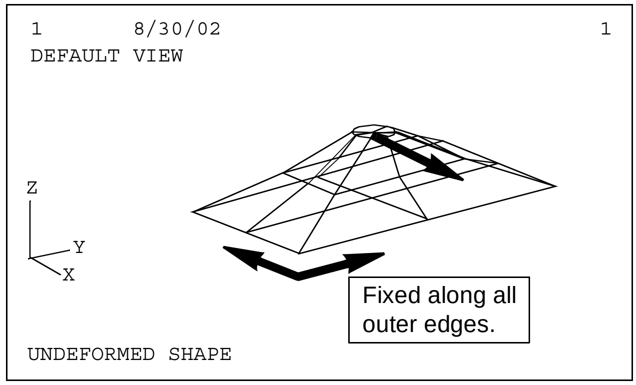
Structural Plotter Examples

This section uses tutorial examples to illustrate the capabilities of the Structural Plotter. Included are examples of both model plots and plots of the solution responses. The examples use a simple model designed to illustrate the plotting concepts.

It is easy to create a model plot using all of the default values:

```
OUTPUT(PLOT)
PTITLE=DEFAULT VIEW
PLOT
```

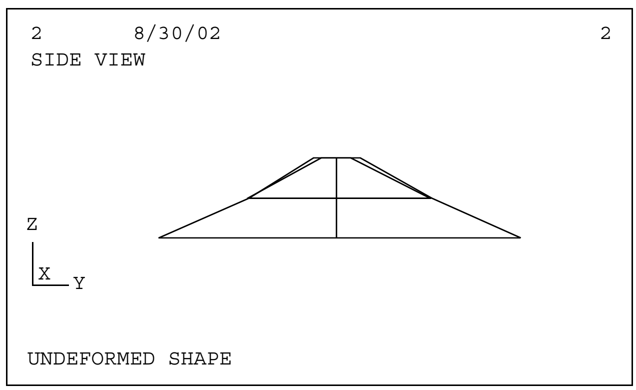
The simple model used for these examples is fixed along the edges of its base and a point load is applied to a single point on the upper ring as shown.



In many cases, model checkout includes plotting the three views of the structure. This can be done several ways:

```
PTITLE=SIDE VIEW
VIEW = 0.,0.,0.
AXES = X,Y,Z
PLOT
```

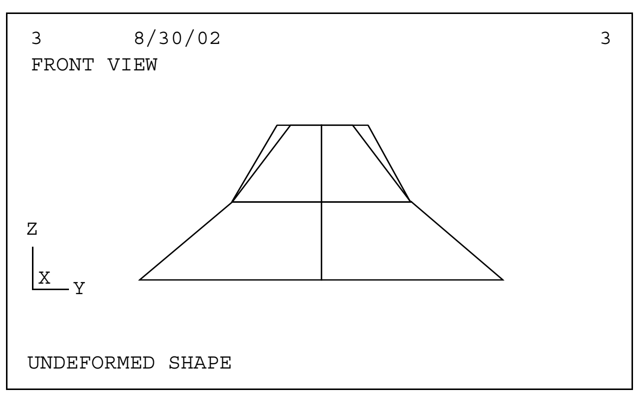
In this case, the VIEW command was used to clear the default viewing angles and the AXES command used to align the model X-Axis with the R-Axis of the plot. You could also accomplish this by using the VIEW command with 90 degree rotations.



A front view is created in a similar manner:

```
PTITLE=FRONT VIEW
AXES = Y,X,Z
PLOT
```

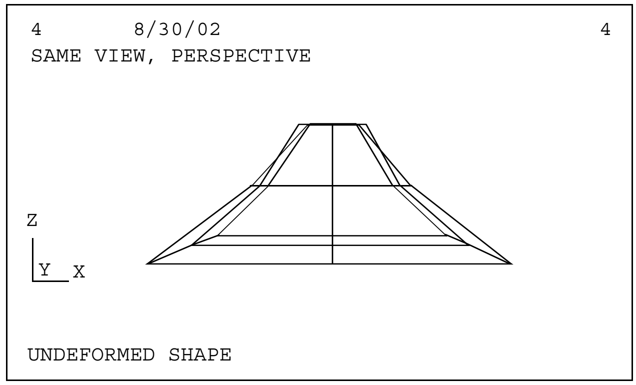
The VIEW command is no longer needed because the previous plot had reset the values. This time, the AXES command is used to align the model Y-Axis with the R-Axis of the plot.



Unless otherwise selected, all plots are created using an ORTHOGRAPHIC PROJECTION. In some cases, you may prefer the realism afforded by the PERSPECTIVE PROJECTION:

```
PTITLE=SAME VIEW WITH PERSPECTIVE
PERSPECTIVE PROJECTION
AXES = Y,X,Z
VIEW = 0.,0.,0.
PLOT
```

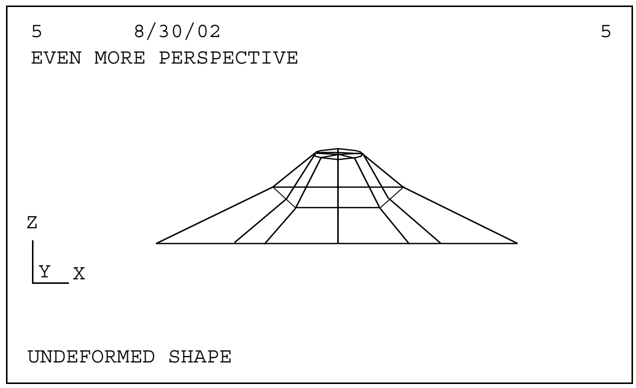
Note that the AXES and VIEW must be reset if the PROJECTION is changed.



You may control the level of perspective distortion. For example:

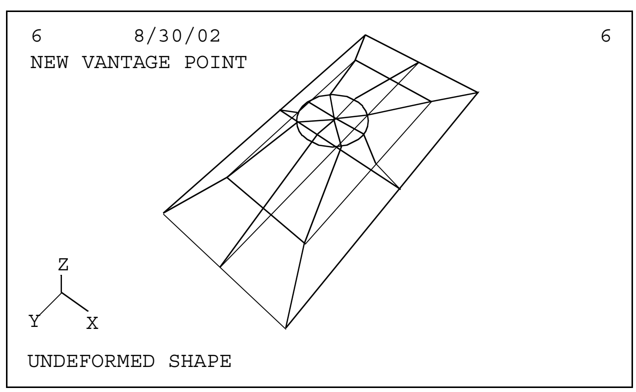
```
PTITLE=EVEN MORE PERSPECTIVE
PERSPECTIVE FACTOR 60.0
PLOT
```

The factor specified is an approximate ratio of the size of the background of the plot to the foreground. In this case, 60.0 indicates that the background is about 60% smaller than the foreground.



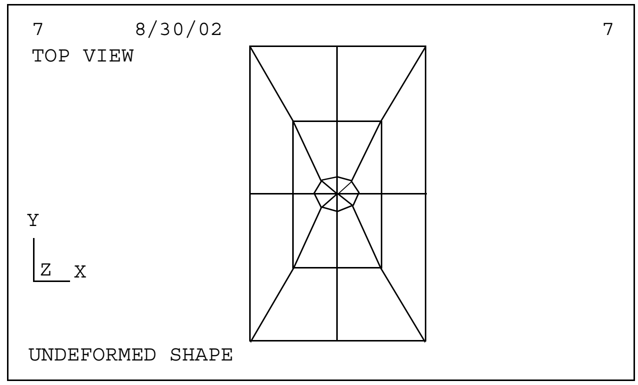
You may move to a different viewing point to better understand your model, for example:

```
PTITLE=NEW VANTAGE POINT
VANTAGE POINT
PLOT
```



Finally, the third view is plotted:

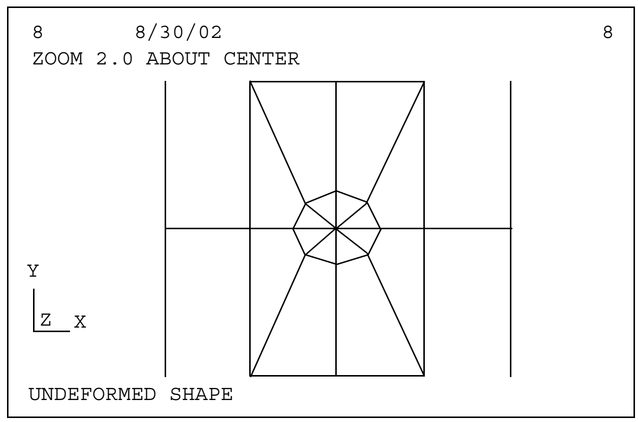
```
PTITLE=TOP VIEW
ORTHOGRAPHIC PROJECTION
VIEW 0.,0.,0
AXES = Z,X,Y
PLOT
```



The factor specified is an approximate ratio of the size of the background of the plot to the foreground. In this case, 60.0 indicates that the background is about 60% smaller than the foreground.

In many cases, you may wish to enlarge your plot about a particular point. For example:

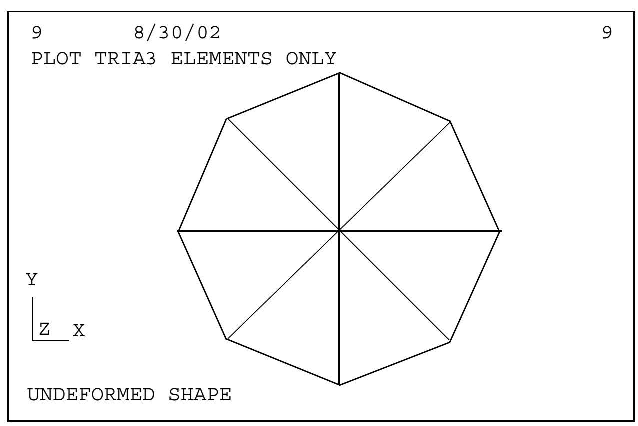
```
PTITLE=ZOOM 2.0 ABOUT CENTER
ZOOM 2.0
PLOT
```



results in the plot shown. Note that the structure was automatically clipped at the boundaries. You may specify any center of ZOOM–you are not limited to the center of the plot. A factor of less than one will result in a reduction in the size of the model.

Now, you wish to create a plot that is just the TRIA3 elements on the top surface. This is easily accomplished with the commands:

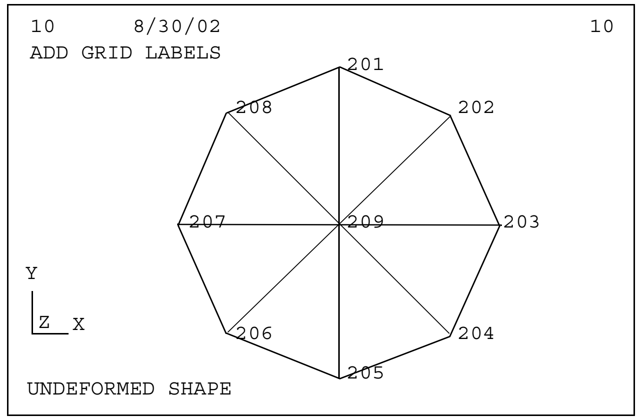
```
SET 2 = TRIA3
PTITLE=PLOT TRIA3 ELEMENTS ONLY
PLOT SET 2
```



Note that all other viewing options have remained in place, and that the selected SET is rescaled to fill the viewing area.

Model checking usually involves identifying GRID points and elements in the model. To add labels for the GRID points, use:

```
PTITLE=ADD GRID LABELS
PLOT SET 2 LABEL GRIDS
```

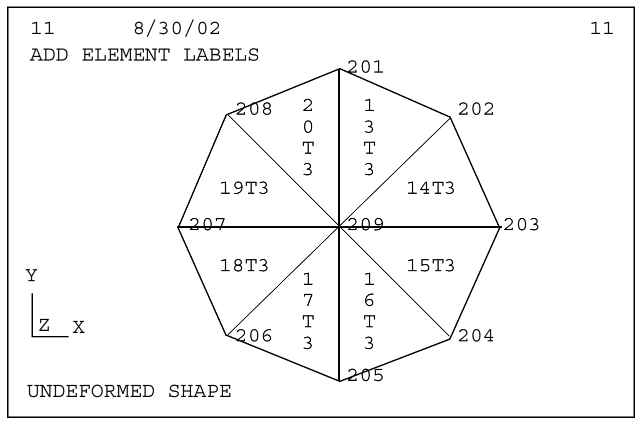


For simple plots like this one, adding the element identification numbers is also useful:

```
PTITLE=ADD ELEMENT LABELS
PLOT SET 2 LABEL BOTH
```

Note that the element type is appended to the identification number as:

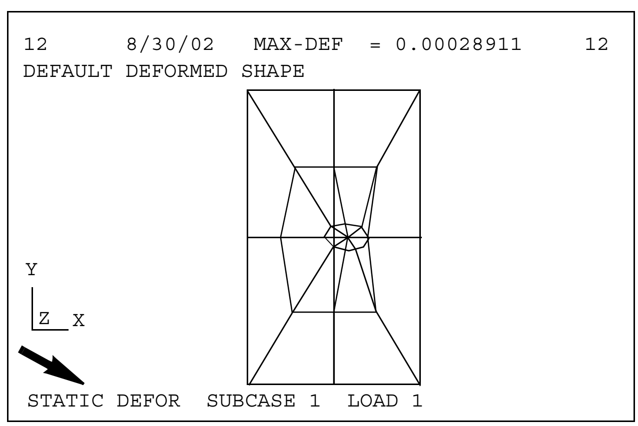
```
16T3
```



Plotting the deformed shape of the structure is equally simple. For example:

```
PTITLE=DEFAULT DEFORMED SHAPE
PLOT STATIC DISP DEFORMED
```

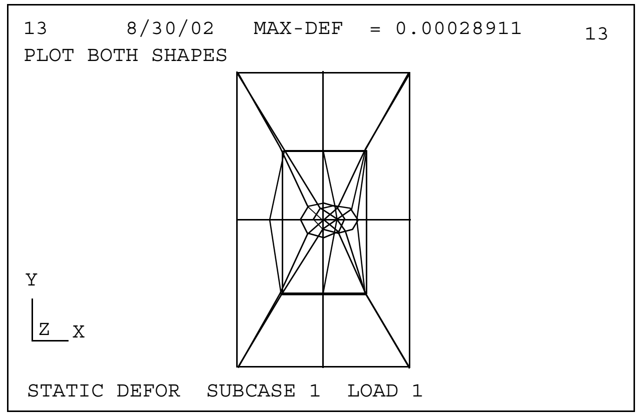
Note that the subcase is automatically noted.



You may also plot the original, or undeformed, shape by using:

```
PTITLE=PLOT BOTH SHAPES
PLOT STATIC DISP UNDEFORMED
     DEFORMED
```

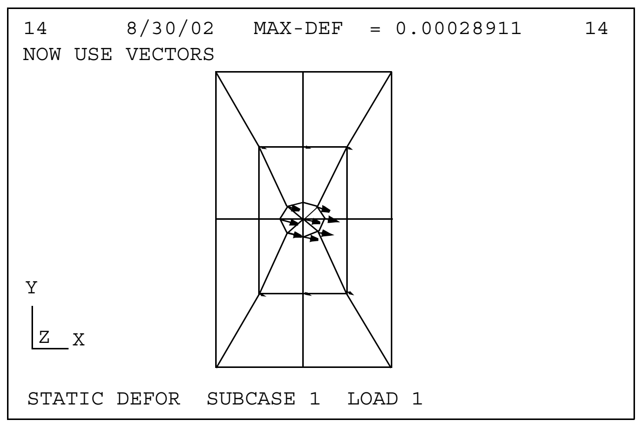
For other than small models, the number of lines in such plots are often excessive. The VECTOR option can help solve this problem.



You may plot vectors which represent the deformations of your model. You may select individual components of displacement or, as in this example, the resultant:

```
PTITLE=NOW USE VECTORS
PLOT STATIC DISP VECTOR R
```

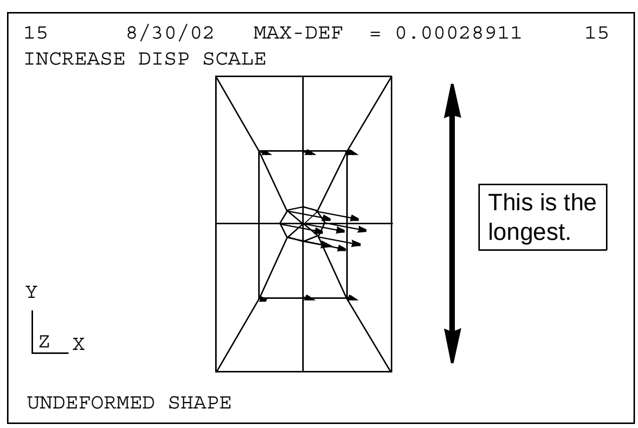
The length of the vectors drawn depends on the scale that you select in the same manner as the deformed shape.



The deformation scale is changed by a single command:

```
PTITLE=INCREASE DISP SCALE
DEFORMATION SCALE 20.0
PLOT STATIC DISP VECTOR R
```

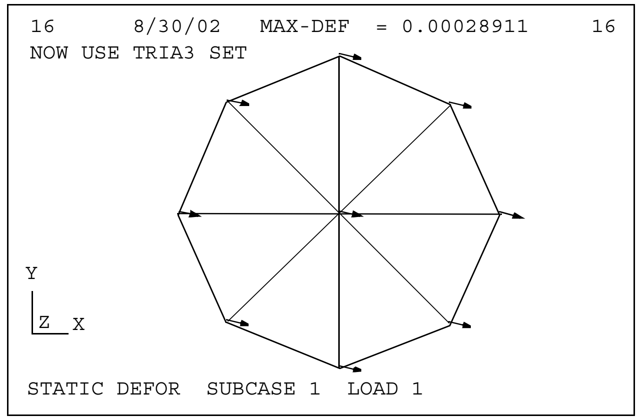
The factor represents a percentage of the maximum dimension in the plotted model. For example, 20.0 indicates that the largest deformation will be plotted such that it has a motion equal to 20% of the longest plot dimension, as noted.



Naturally, deformed plots may also be created for sets:

```
PTITLE=NOW USE TRIA3 SET
PLOT STATIC DISP SET 2 VECTOR R
```

The length of the vectors drawn depends on the scale that you select in the same manner as the deformed shape.



6 XY PLOTTING

Introduction

When you use the dynamic response analysis disciplines within NASTRAN-CORE, most often you are faced with reviewing and analyzing large volumes of solution results. To aid you in this task, NASTRAN-CORE provides an XY plotting capability for representing this data. The actual XY plots, like the Structural Plots, are not created during your NASTRAN-CORE execution. Rather, the plot information is placed on a file that is postprocessed.

NASTRAN-CORE generates two plot files, .PLT1 and .PS, when plots are requested. The PLT1 file, a binary plot file, can be interpreted by a user-modifiable NASPLOT program included in the NASTRAN-CORE deliverable. The PS file is readily printable by any Postscript printer.

In some cases, you must contact your NASTRAN-CORE Support Specialist for details describing how your site interfaces with this postprocessor.

This section describes the XY Plotter terminology, the input data requirements for creating such plots, and provides examples which illustrate the use of many of the plotter commands.

XY Plotting Terminology

Most frequently, an XY Plot consists of one or more functions of the form, $y_i = f_i(x)$. The independent variable, x, may be: time, if you are solving transient response problems; frequency, when you are solving frequency response problems; or subcase identification number when you are solving either linear or nonlinear statics problems. The dependent variables, y_i , may represent a response quantity that varies with the independent variable. Such responses include displacement, accelerations, and the many other input and solution quantities which are available. This type of XY Plot is called a *history* plot.

Correlation Plot. This type of plot allows you to plot one response quantity against another. For these plots, the set of XY pairs is extracted from the data, sorted by increasing value of the X coordinate, and then plotted.

Frame. Analogous to photography, each displayable plot is called a frame.

Whole Frame Plot. Each frame may contain one or two plots. When the frame contains a single plot, it is called a whole frame plot.

Half Frame Plots. Alternately, a frame may contain two plots, a *top half frame plot* and a *bottom half frame plot*.

Select between these two options based on the plot resolution that you require and the use that you intend for the graphic results. Obviously, the two half frame plots will not allow the accuracy or readability that a whole frame plot will. On the other hand, half frame plots are perfectly fine for a fast, qualitative analysis of the data.

Curves. Each of the plots you request may contain one or more curves. The curves may be drawn as a set of points which may optionally be connected by straight line segments. Additionally, graphic symbols may be placed at the locations of the actual data points. Symbols change from curve to curve so that you may differentiate them.

There are many optional commands that you may use to modify the format and labeling of your plots. These commands operate on *plot elements*. **Most of these commands function as toggles: you turn them on or off and they remain in that state until you toggle them again.**

Command Verbs. The XY Plotter includes a number of different functions which are selected by specifying a command verb. Available commands not only result in graphic plots, but allow you to print the coordinates of the plotter points, obtain a summary of the minimum and maximum values for each curve, write the plotter coordinates on a file for use by other programs, and allow you to create plots on a standard line printer in the event that you do not have access to a graphics device.

All of these options and capabilities are discussed in the remainder of this section.

XY Plotter Data Requirements

This section provides you with a description of the NASTRAN-CORE input data required for creating XY Plots.

Executive Control Commands

Although there are no specific Executive Control commands which are required to perform XY Plotting,

Both XY Plots and Structural Plots are written to the same plot file.

Substructure Control Commands

The substructuring capability has no specific control commands to support XY Plotting. You may create XY Plots during Phase 3 data recovery operations in the same manner as you create them for non-substructuring analyses.

Case Control Commands

Table 6-1 As was the case for the Structural Plotter, the XY Plotter commands form a subpacket within the Case Control command packet as shown in **Figure 6-1**.

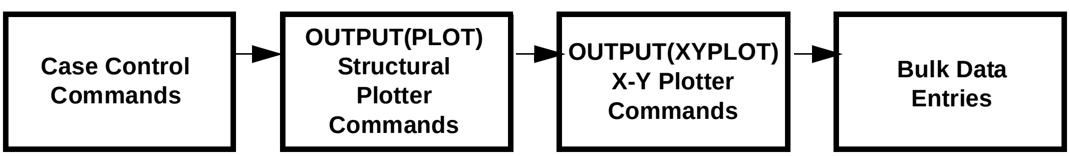


Figure 6-1 Location of the XY Plotter Subpacket

Both the Structural Plotter subpacket and the X-Y Plotter subpacket may be interchanged in the data stream. It is not necessary for you to issue Case Control output request commands when you are requesting XY Plots. NASTRAN-CORE automatically extracts the necessary data from the solution results.

Bulk Data

There are no Bulk Data entries which control XY Plots in any manner. Naturally, many entries are used to control the domain of the solution results which are computed and available to create the plots.

The XY Plot Elements

The following sections describe the XY Plot elements. Each of these elements has a name which corresponds to the XY Plotter command used to control it. These commands are different depending on whether you are creating whole frame or half frame plots. However, the similarity of these commands is such that the term command family is used to describe them. For example, a command called COMM when applied to whole frame plots is called XCOMM if it acts only on an X-Axis plot element, and it is called YCOMM if it acts on the corresponding Y-Axis plot element. Similarly, if the command may also be applied to upper and lower half frame plots, the command names would be XTCOMM, YTCOMM, and XBCOMM, YBCOMM, respectively. This will be made clearer in the descriptions in the following sections.

Whole Frame Plots

Figure 6-2 shows a typical whole frame plot. It illustrates the most often encountered plot elements. Notice that both the X-Axis and the Y-Axis have text labels. These elements are called the XTITLE and YTITLE, respectively. Unless you request otherwise, solid lines are drawn representing the axes themselves. They are naturally called the XAIS and YAXIS. The two axes intersect at the XINTERCEPT and YINTERCEPT. In the figure, the default intercept coordinates of 0.0 and 0.0 were used. Notice that there are small cross-hairs, called tics, at various positions along both axes. These are called LEFT TICS and LOWER TICS (or BOTTOM TICS). The tics may be toggled ON or OFF, as you wish. The number of tics which appear on the axes are called XDIVISIONS and YDIVISIONS. Because NASTRAN-CORE creates axis labels which are rounded to avoid the use of irregular values, the number of divisions that are actually plotted may vary slightly from the number specified. For example, the default value of five divisions has been used for the plot in , but only four divisions appear so that the axis labels are round.

The plot contains two curves, each of which is connected by a LINE. Optional SYMBOLS have been placed at actual plot point locations. Finally, the frame itself has been given a title, called an FTITLE, which is included along with the standard Case Control titles.

Half Frame Plots

In addition to the whole frame plot elements discussed above, there is another group of elements which are found in half frame plots. **Figure 6-3** shows such a plot. Again, notice that the X-Axis and the Y-Axis have textual labels for both the upper and lower plots. While the label on the X-Axis is still the XTITLKE, the Y-Axis label for the top plot, YTTITLE, and bottom plot, YBTITLE, can be independently defined. As with whole frame plots, solid lines are drawn representing the axes themselves unless you request otherwise. The Y-Axis is still the YAXIS, but the X-Axes are now the XTAXES and XBAXIS. Also, separately controllable are: the Y-Axis intersection, YTINTERCEPT and YBINTERCEPT; the left tics, TFLEFT TICS and BLEFT TICS; and many other plot elements. These are discussed in detail in the next section.

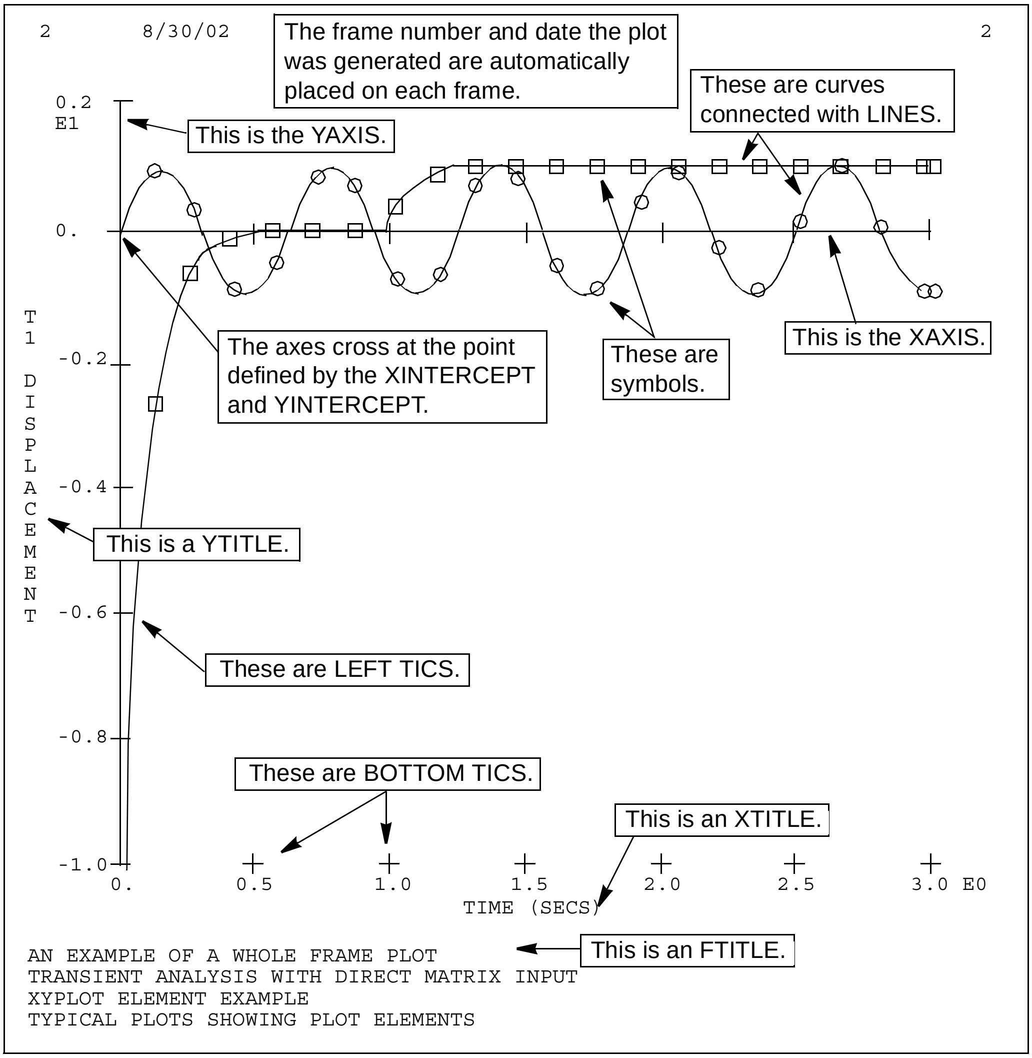


Figure 6-2 Plot Elements for Whole Frames

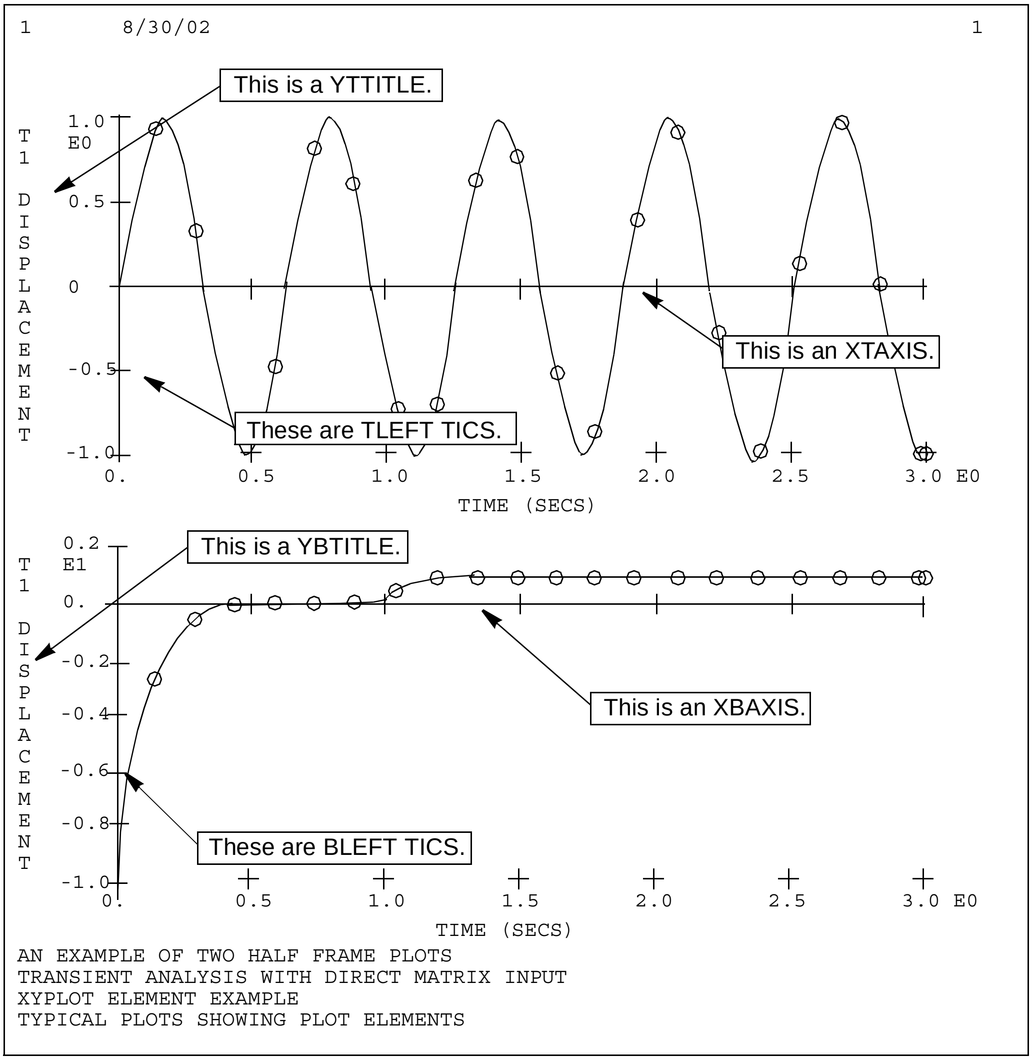


Figure 6-3 Plot Elements for Half Frames

XY Plotter Commands

The general form of the XY Plotter Command packet is:

```
OUTPUT(XYPLOT)
   option_command
   option_command
   ...
   PLOT_command_verb
   option_command
   option_command
   PLOT_command_verb
   ...
OUTPUT(PLOT) or BEGIN BULK
```

The packet must begin with the OUTPUT(XYPLOT) command and it ends when either an OUTPUT(PLOT_ or BEGIN BULK command is encountered. The other commands in the packet define the plot frames that you wish to create. Some of these are option_commands which define the characteristics of the plots to be made while others are the PLOT_command_verbs which actually specify which is to be plotted or printed.

Note: When performing Multidisciplinary Design Optimization, only a single XY Plotter Command packet is used. The requested plots are generated for each appropriate discipline at each design iteration.

The available XY Plotter Commands are summarized in **Table 6-2** where they are grouped in functional categories. The remainder of this section will provide you with an overview of these commands.

Table 6-2 XY Plotter Commands

Commands Which Control:	When Selecting Plots Which Are:		
	Whole Frame	**Upper Half Frame**	**Lower Half Frame**
Data Range Selection	XMIN XMAX YMIN YMAX	YTMIN YTMAX	YBMIN YBMAX
Axis Selection and Control	XAXIS YAXIS	XTAXIS	XBASIX
	XDIVISIONS YDIVISIONS	YTDIVISIONS	YBDIVISIONS
	XINTERCEPT YINTERCEPT	YTINTERCEPT	YBINTERCEPT
Grid Lines	XGRID YGRID	XTGRID YTGRID	XBGRID YBGRID
Titling	PTITLE	PTITLE	PTITLE
Scale Markers and Values	UPPER SCALES LOWER SCALES LEFT SCALES RIGHT SCALES XVALUE SCALES YVALUE SCALES	TLEFT SCALES TRIGHT SCALES YTVALUEL SCALES	BLEFT SCALES BRIGHT SCALES YBVALUE SCALES

Table 6-2 XY Plotter Commands (continued)

Commands Which Control:	When Selecting Plots Which Are:		
	Whole Frame	Upper Half Frame	Lower Half Frame
Curve Appearance	DRAWLINE	DRAWLINE	DRAWLINE
	SYMBOL	SYMBOL	SYMBOL
	LINESTYLE	LINESTYLE	LINESTYLE
Logarithmic Scales	XLOG YLOG	YTLOG	YBLOG
Utility	CLEAR	CLEAR	CLEAR
Graphics Device	ASPECT RATIO	ASPECT RATIO	ASPECT RATIO
	CHARACTER PRECISION	CHARACTER PRECISION	CHARACTER PRECISION
	CHARACTER SCALE	CHARACTER SCALE	CHARACTER SCALE

Data Range Selection. The command families MIN and MAX allow you to specify the minimum and maximum values of the data that you wish to plot. This allows you to isolate active response areas in the plot, thus making the data more readable. Note that you may only control the minimum and maximum value of the ordinate when using half frame plots, because both plots must share the same X-Axis data range.

Axis Selection and Control. You may select the drawing of the X- and Y-Axes on your plots by using the AXIS command family. The approximate number of divisions along each axis may be selected with one of the DIVISIONS family of commands. You may also control the point at which the axes intersect with the INTERCEPT commands.

Grid Lines. You may simulate graph paper by using the GRID command family to draw lines along the divisions of the coordinate axes. This option is best used if you plan to read values from the plot. Generally, plots are more legible with fewer lines.

Titling. You may add an additional title to your plot frame by using the PTITLE command. The other TITLE commands allow you to label the coordinate axes. Note that both X-Axes on half frame plots must have the same title.

Scale Markers and Values. You may also request that tic marks and scale values be drawn on the upper, lower, left and right borders of each plot frame by using the SCALES command family. These commands also allow you to control which tic marks will have scale values.

Curve Appearance. You may draw curves which place a symbol at each data point by using the SYMBOL command. The DRAWLINE command is used to connect the data points with straight line segments. Naturally, you may select both of these options. Depending on the capabilities of your plot postprocessor, you may also use the LINESTYLE command to control the appearance of the straight line segments.

Logarithmic Scales. The default XY Plotter mode is to plot data using cartesian coordinates. You may use the LOG command family to use semi-log or log-log scales. If you select logarithmic scales, then the XY Plotter automatically determines the number of cycles to plot and control the labeling of the axes.

Utility Operation. The CLEAR command may be used selectively to enable or disable various plot options.

Controlling the Graphics Device. There are three commands that you use to define the characteristics of the graphics device that will be used to create your plots using the sasplot program or an alternate program developed at your site. Contact your NASTRAN-CORE Systems Support Specialist to get information on these parameters.

The XYPLOT Command

Use the XYPLOT command to create histograms of input data and solution results as a function of time or frequency depending on the type of analysis you are performing. You may also use it to process random response results by creating power spectral density and autocorrelation plots. Finally, you may create correlation plots of one response quantity against another using time or frequency as a parameter. The XYPLOT command results in the creation of a plot file that is then postprocessed to display the plot on your graphics device.

The general form of the XYPLOT command family is:

```
{ XYPLOT  }
{ XYPRINT }  { HISTORY     }
{ XYPUNCH }  { CORRELATION }  y_data VS x_data ptype subcase_list curve_list
{ XYPEAK  }
{ XYPAPER }
```

Any combination of the plot command verbs may be used in a single plot command. There are many options within each clause of these commands. All of these clauses are the same for each command, only the form of output differs, as you will see in subsequent sections. Note that when you create history plots, the HISTORY and VS keywords and the x_data selection are not required.

Selecting the y_data and x_data. The first data included on the command is a definition of the y_data and x_data that will form the plot. Various parameters and response quantities may be selected for plotting for each axis.

The plot_type. There are three plot_types from which you may choose:

```
{    RESPONSE     }
{ AUTOCORRELATION }
{      PSDF       }
```

The default plot_type is RESPONSE. This indicates that you are plotting one of the solution response quantities or input quantities. The other two options, AUTOCORRELATION and the power spectral density function, PSDF, may only be used if you are performing a random analysis.

The subcase_list. The subcase_list is simply a list of the subcases for which you wish plots to be generated. The general form of this list is:

```
subid_1 [,subid_2,...]
```

where each subid is simply the identification number of a SUBCASE that you have defined in your Case Control command packet.

The curve_list. The curve_list defines the number of frames, whether plots will be whole frame or half frame format, and the output quantities that will be plotted. Each frame specification entry begin with a slash (/) as shown below:

```
/ frame_1 [/ frame_2 ...]
```

Note that any number of frames may be defined. Each of the frame descriptors is composed of a list of curve descriptors:

```
curve_1 [, curve_2, ...]
```

Each of these descriptors requests a single curve and specifies whether the curve will be placed on a whole frame or on a top or bottom half frame. A whole frame curve is defined by:

```
grid_id ( comp ) , ... or
elem_id ( comp ) , ...
```

First, you must specify a grid_id or elem_id whose response component is to be plotted. Then, enclosed in parentheses, you select the code for that response component. To define half frame curves, the form is:

```
grid_id ( [comp_top] [, comp_bot]) , ... or
elem_id ( [comp_top] [, comp_bot]) , ...
```

The first component named comp_top will be plotted in the upper half frame. Similarly, the second quantity, comp_bot, will be plotted in the lower half frame. Either of these quantities may be omitted in which case there will be no curve on the corresponding half frame plot. The comma must always appear in the command to differentiate the two curves. Note that you may change response components, grid_ids, and elem_ids from curve to curve.

XYPLOT Output. In addition to creating the plot file used by your graphics postprocessor, the XYPLOT and, in fact, all of the XY Plotter command verbs, generates a summary for each curve which includes the quantities plotted, the titling information, and the ranges of the data values. An example is shown in **Table 6-3**.

Table 6-3 XY Plotter Curve Summary

```
                                       XY-OUTPUT SUMMARY
SUBCASE          1
RESPONSE
HISTORY CURVE, DISPLACEMENT RESP 4(T3) VS TIME

XY-PAIRS WITHIN FRAME LIMITS WILL BE PLOTTED.
PLOTTER SPECIFIED IS CRT

THIS IS CURVE    1 OF WHOLE FRAME    1

CURVE  TITLE = WHOLE FRAME, SINGLE CURVE, ALL DEFAULTS

X-AXIS TITLE =

Y-AXIS TITLE =

THE FOLLOWING INFORMATION IS FOR THE ABOVE DEFINED CURVE ONLY.

WITHIN THE FRAME X-LIMITS       ( X = 0.000000E+00 TO X =    8.000001E-01 )

                         THE SMALLEST Y-VALUE =   -5.000000E-02 AT X =     0.000000E+00
                         THE LARGEST  Y-VALUE =    2.839540E-02 AT X =     1.600000E-02

WITHIN THE X-LIMITS OF ALL DATA ( X = 0.000000E+00 TO X =    8.000001E-01 )

                         THE SMALLEST Y-VALUE =   -5.000000E-02 AT X =     0.000000E+00
                         THE LARGEST  Y-VALUE =    2.839540E-02 AT X =     1.600000E-02

                                        END OF SUMMARY
```

The XYPRINT Command

Use the XYPRINT command to create tables of the actual data values that are used or would be used to create a corresponding XYPLOT. This feature must be used when performing random response analysis because it is the only method available for printing solution data. **Table 6-4** illustrates a sample output from the XYPRINT command.

Table 6-4 Example of XYPRINT Output

HISTORY CURVE, DISPLACEMENT RESP 2(T3) VS TIME - WHOLE FRAME

STEP	X-DATA	Y-DATA
1	0.000000E+00	0.000000E+00
2	2.000000E-03	-3.798743E-04
3	4.000000E-03	-4.968450E-03
4	6.000000E-03	-8.386047E-03
5	8.000000E-03	-4.311383E-03
6	1.000000E-02	4.382742E-03
7	1.200000E-02	6.099680E-03
8	1.400000E-02	6.316201E-04
9	1.600000E-02	-2.237488E-04
10	1.800000E-02	5.604031E-03
11	2.000000E-02	8.317089E-03
12	2.200000E-02	4.415992E-03
13	2.400000E-02	2.026191E-04
14	2.600000E-02	-2.814618E-04
15	2.800000E-02	5.873423E-04
16	3.000000E-02	-3.714329E-04
17	3.200000E-02	-3.283890E-03
18	3.400000E-02	-5.641322E-03
19	3.600000E-02	-5.226959E-03
20	3.800000E-02	-3.133581E-03

XYPUNCH - Interfacing with Postprocessors

You may use the XYPUNCH command to write the X-Values and Y-Values of each curve to a file you have assigned with a USE=PUNCH. The data for each curve is preceded by a series of descriptor records, each of which begin with a dollar sign, $. The actual curve data then follows, one record for each pair of XY values. Each record contains a sequence number in positions 72 through 80. This feature is useful for importing these data into another program. The Fortran format of these records is:

```
FORMAT(2E20.6,32X,I8)
```

Table 6-5 shows a sample of the contents of the resulting file.

Table 6-5 Example of XYPUNCH Output

```
$TITLE    = SIMPLE TRANSIENT RESPONSE ANALYSIS                           1
$SUBTITLE = USER'S GUIDE SAMPLE PROBLEM                                  2
$LABEL    = USING VARIOUS PLOT ELEMENTS AND OPTIONS                      3
$SUBCASE ID =       1                                                    4
$XY HISTORY CURVE                                                        5
$XY DISPLACEMENT RESP 2(T3) VS TIME                                      6
        0.000000E+00       0.000000E+00                                  7
        2.000000E-03          -3.798743E-04                              8
        4.000000E-03          -4.968450E-03                              9
        6.000000E-03          -8.386047E-03                             10
        8.000000E-03          -4.311383E-03                             11
        1.000000E-02           4.382742E-03                             12
        1.200000E-02           6.099680E-03                             13
        1.400000E-02           6.316201E-04                             14
        1.600000E-02          -2.237488E-04                             15
        1.800000E-02           5.604031E-03                             16
        2.000000E-02           8.317089E-03                             17
        2.200000E-02           4.415992E-03                             18
        2.400000E-02           2.026191E-04                             19
        2.600000E-02          -2.814618E-04                             20
        2.800000E-02           5.873423E-04                             21
        3.000000E-02          -3.714329E-04                             22
```

XYPEAK - Data Range Scanning

The XYPEAK command allows you to scan the data range of the requested solution results and to print a summary of the maximum and minimum values of each curve that you draw. **Table 6-6** shows a sample of the output resulting from this command. You obtain a single table that contains all of the data for all of the XYPEAK commands that appear in your XY plotter command subpacket.

Table 6-6 Example of XYPEAK Output

```
                                       XY-OUTPUT SUMMARY

                            MAXIMUM AND MINIMUM VALKUES OF THE DATA
            X LIMITS FOR THIS TABLE     MAX =  2.000000E-01     MIN =  0.000000E+00

         POINT OR COMPONENT VECTOR PLOT   MAXIMUM   CORRESPONDING     MINIMUM   CORRESPONDING
SUBCASE ELEMENT ID  NUMBER   TYPE  TYPE      Y            X              Y             X

   1        2         T3     DISP  RESP 8.317089E-03 2.000000E-02 -8.386047E-03 6.000000E-03
```

The XYPAPER Command

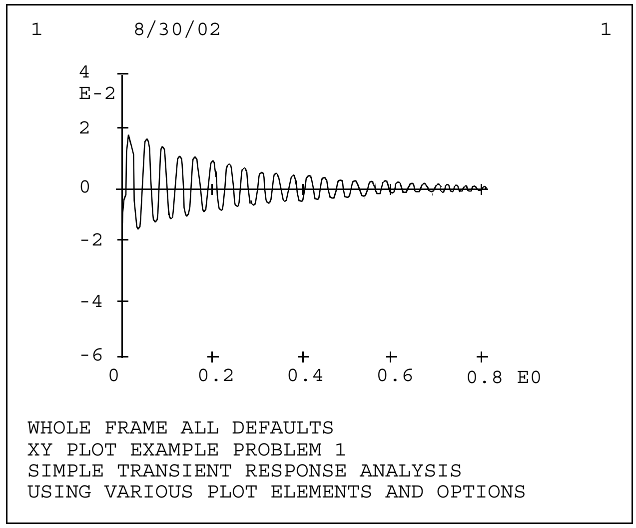
In the event that your computer site does not have graphics hardware suitable for displaying XY Plots, you may use the XYPAPER command. This command generates the plots on your line printer. The Y-Axis is plotted in 132 columns across the width of the paper and the X-Axis is plotted from page-to-page in an open-ended fashion. Only symbols are placed on these XYPAPER plots. The first curve on the frame uses the symbol *. Successive curves are denoted by the symbols O, A, B, C, D, E, F, G, and H. Although the resolution of these plots is low, they still may provide important qualitative information.

XY Plotter Examples

This section uses tutorial examples to illustrate the capabilities of the XY Plotter. The examples are drawn from two models. The first is a transient response analysis of a simple cantilever beam model which is excited by an initial displacement. The second is a material nonlinear analysis used to illustrate the correlation plot.

You may create an XY Plot very simply if you select all the standard default values. The plot on the right was created with the commands:

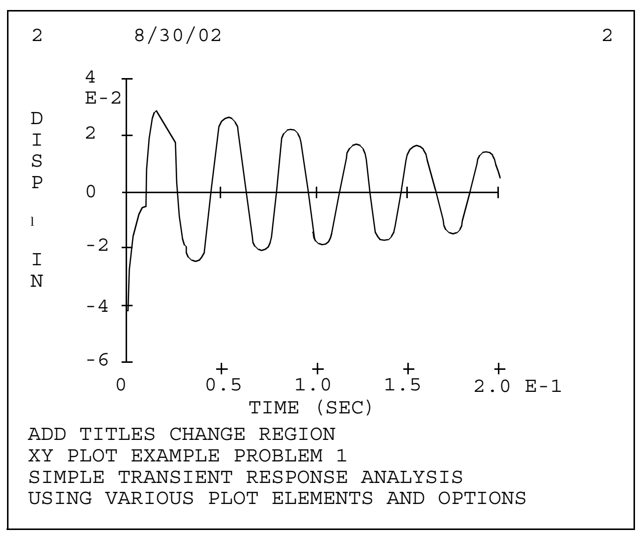
```
OUTPUT(XYPLOT)
PTITLE=WHOLE FRAME ALL DEFAULTS
XYPLOT DISP RESPONSE / 4(T3)
```



Although the displacement results are well represented, usually you will add titling to your plot. Often, you will also want to zoom in on a smaller region of interest. Use the commands:

```
XTITLE=TIME(SEC)
YTITLE=DISP * IN
FTITLE=ADD TITLES, CHANGE REGION
XMAX = 0.2
XYPLOT DISP RESPONSE / 4(T3)
```

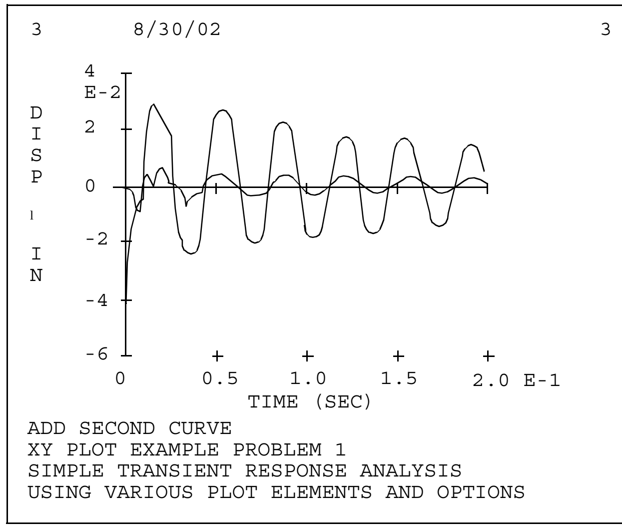
The resulting plot is much more understandable.



You may often wish to compare several responses by plotting them on the same graph. This is done simply by specifying them in a list. For example, to add the T3 displacement for GRID 2, you could use the commands:

```
FTITLE=ADD SECOND CURVE
XYPLOT DISP RESPONSE / 4(T3),2(T3)
```

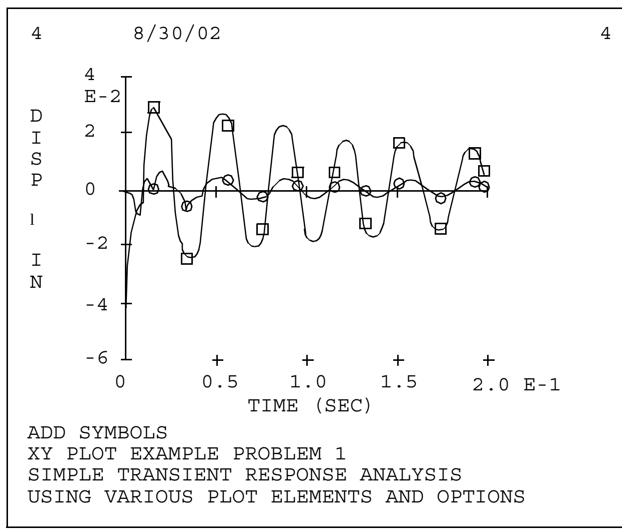
Notice that all of the titling and the plot region are still in effect for this plot.



More than one plot is difficult to interpret. So, you may place different symbols on each curve:

```
FTITLE=ADD SYMBOLS
SYMBOL 6,10
XYPLOT DISP RESPONSE / 4(T3),2(T3)
```

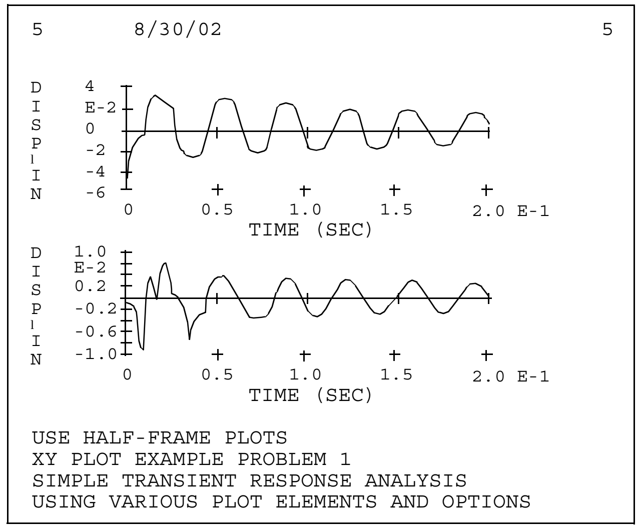
The SYMBOL command selects the type of the first symbol. The others are then selected using the rules described in the User's Reference Manual. The value 10 indicates that a symbol will be placed every ten data points.



Your plots may be more readable if you plot responses as separate upper and lower half frames. The example plot was created with the commands:

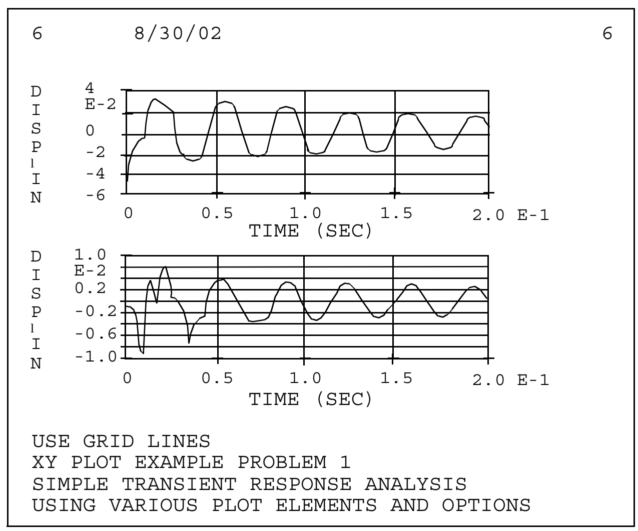
FTITLE=USE HALF-FRAME PLOTS
XYPLOT DISP RESPONSE / 4(T3),2(,T3)

Now, each response is plotted individually. The smaller plots have less accuracy than the whole frame plots.



You may also add grid lines to your plots to simulate graph paper. The example was created with the commands:

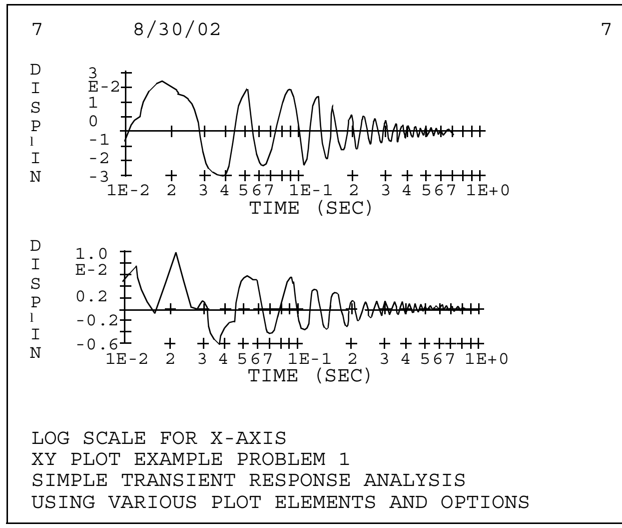
FTITLE=ADD GRID LINES
XGRID ON
YGRID ON
XYPLOT DISP RESPONSE / 4(T3,),2(,T3)



You may also select logarithmic scales in one or both coordinate directions. This is often used in transient response analyses. The example was created with:

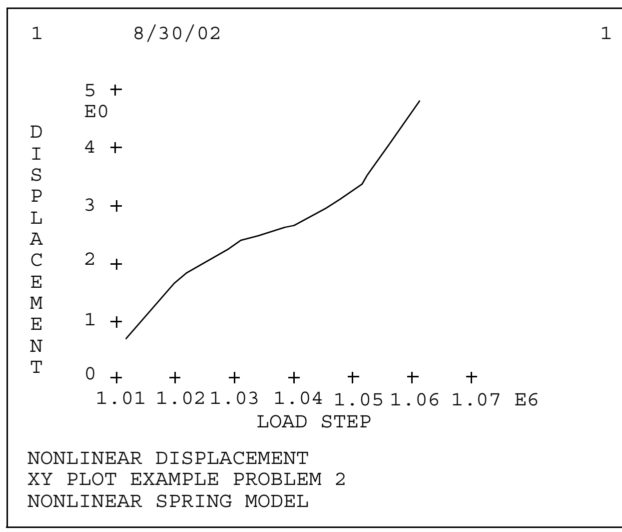
```
FTITLE=LOG SCALE FOR X-AXIS
XLOG ON
XYPLOT DISP RESPONSE / 4(T3,),2(,T3)
```

Note that when you use log scales, NASTRAN-CORE automatically selects the number of cycles. You may not control this value unless you reduce the data range requested.



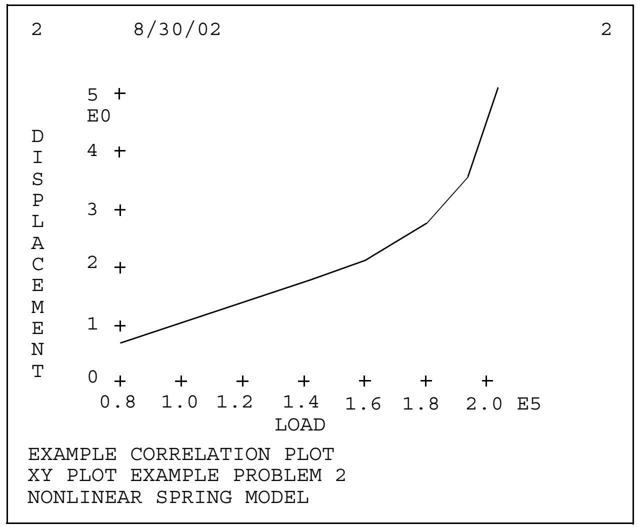
The second example problem is the non-linear analysis of a simple spring system. The first plot shows the displacement as a function of the nonlinear load step. It was created with the commands:

```
FTITLE=NONLINEAR DISPLACEMENT
XTITLE=LOAD STEP
YTITLE=DISPLACEMENT
XYPLOT DISP RESPONSE / 1(T1)
```



Since one is often interested in the force-deflection results for nonlinear analyses, a correlation plot can be created with the command:

```
FTITLE=EXAMPLE CORRELATION PLOT
XTITLE=LOAD
XYPLOT CORRELATION
       DISP VS OLOAD/
       1(T1) VS 1(T1)
```



7 HEAT TRANSFER PROBLEMS

Introduction

NASTRAN-CORE heat transfer capability can be used as a separate analysis to determine temperatures and fluxes or to determine temperature input values for structural problems, Steady and transient problems can be solved, including heat conduction with variable conductivity for static analysis, film heat transfer, and nonlinear fourth power law radiation.

The NASTRAN-CORE heat transfer model definition is similar to that used for the structural model. The same grid points, coordinates system definitions, the use of elements, single and multi point constraints can be used for both problems. There are several differences, however, including:

1. Only a single degree of freedom, the temperature, is associated with a Grid point,
2. The specification of loads
3. The specification of boundary film heat conduction
4. The specification of non linear elements

The only variable in heat transfer problems is the temperature at the grid points as compared to the six degrees of freedom associate with the structural model. Compared to the structural model, additional Grid or Scalar points are introduced for fluid ambient temperature in convective heat transfer. When radiation is included or the conductivity is temperature-dependent then the problem becomes non linear.

The heat transfer model in NASTRAN-CORE is compatible with the structural model. If the same finite elements are appropriate for both models then the same Grid points and elements can be used for both analyses. As is the case in structural analysis the choice of the finite element mesh is left to the analyst.

Heat transfer analysis in NASTRAN-CORE uses many of the same Bulk Data entities used in structural analysis and which are described in Chapter 4 of this manual, including:

Table 7-1 Bulk Data Common to Heat Transfer and Structural Models

Bulk Data Name	Description
CBAR, PBAR	Linear connectivity Element
CBEAM, PBEAM	Linear connectivity Element
CDAMP1, CDAMP2, CDAMP3, CDAMP4 ,PDAMP	Linear capacitance Element
CELAS1, CELAS2, CELAS3,CELAS4, PELAS	Linear connectivity Element
CHEXA, PSOLID	Eight-node solid element
CONROD	Linear connectivity Element
CPENTA, PSOLID	Six-nods solid element
CQUAD4, PSHELL	Four-node shell element
CROD, PROD	Linear connectivity Element
CTETRA, PSOLID	Four-node solid element

Table 7-1 Bulk Data Common to Heat Transfer and Structural Models

Bulk Data Name	Description
CTRIA3, PSHELL	Three-noted shell element
CVISC, PVISC	Linear capacitance Element
DAREA	Applies thermal flux to a node point
DELAY	Time delay in dynamic solutions
DLOAD	Collects flux sets in dynamic analyses
DMI	Direct Matrix Input, must be included in analysis by DMAP Alter
DMIG	Direct Matrix Input at a Grid point. Generally used in dynamics
EPOINT	Extra Point
GRDSET	Specifies default values for Grid
GRID	Specifies spatial location of a node point
LOAD	Collects flux sets for static analysis
MPC	Specifies linear constraint values
MPCADD	Collects sets of MPC
NOLIN1, NOLIN2, NOLIN3, NOLIN4, NOLIN6	Specifies non linear fluxes
OMIT, OMIT1	Specifies matrix partitioning
PARAM	Identifies a PARAM for thermal model, see Chapter 7 of this manual
PLOTEL	Used for creating plots
SLOAD	Scalar load
SPC	Specifies temperature constraints at a node point
SPCADD	Collects sets of SPC
SPOINT	Scalar point
TABLED1, TABLED2, TABLED3 ,TABLED4	Defines a tables associated with time variance of thermal flux
TEMP	Specifies temperature
TF	Transfer function
TLOAD1, TLOAD2	Specifies time-dependent thermal flux field
TSTEP	Specifies time step for dynamic analysis

Heat Transfer Element

The basic elements that generate both conductivity and capacitance element matrices have the same element names as elements used in structural models as shown by the following table:

Table 7-2 Heat Transfer Elements

Element Type	Elements
Linear	CBAR, CBEAM,CROD, CONROD
Surface	CQUAD4, CTRIA3
Solid of Revolution	TRIARG, TRAPRG
Solid	CHEXA, CPENTA, CTETRA
Scalar	CELAS1, CELAS2, CELAS3, CELAS4 CDAMP1, CDAMP2, CDAMP3, CDAMP4

where:

1. The linear elements have a constant cross sectional property
2. The offset on the BAR and BEAM is treated as a perfect conductor
3. The bending characteristics of shell elements as defined by the PSHELL entry is ignored
4. The membrane thickness of shell elements as defined on the PSHELL is the thickness use for conduction
5. The ELAS elements are used in transient analysis to specify temperature constraints
6. The DAMP elements are used to define thermal capacitance
7. Thermal gradients and thermal fluxes are requested using the ELFORCE Case Control command

Thermal Material Properties

Material properties that are referenced on element property Bulk Data must reference Thermal Material Property Bulk data as described by MAT4 and MAT5 Bulk Data that are described below.

The heat capacity, CP, is the product of the material density and the heat capacity per unit mass:

$$CP = \rho C_p$$

Lumped conductivities and thermal capacitance may be defined by ELASi and DAMPi elements respectively

MAT4 - *Thermal Material Property Definition*

Description

Defines the thermal properties for temperature-independent, isotropic materials.

Format

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP						

Example

MAT4	23	.6	.2						

Field	Description
MID	Material identification number. (Integer > 0)
K	Thermal conductivity or convective film coefficient. (Real ≥ 0.0)
CP	Thermal capacitance per unit volume or film capacitance per unit area. (Real ≥ 0.0 or blank)

Remarks

1. The material identification number must be unique with respect to all other MAT4 or MAT5 Bulk Data.
2. If an HBDY Bulk Data entity references the MAT4, K is the conductive film coefficient and CP is the thermal capacitance per unit area.
3. The**MATT4** is used in conjunction with a MAT4 to defined temperature-dependent properties.

MAT5 - *Thermal Material Property Definition*

Description

Defines the thermal properties for temperature-independent, anisotropic materials.

Format

1	2	3	4	5	6	7	8	9	10
MAT4	MID	KXX	KXY	KXZ	KYY	KYZ	KZZ	CP	

Example

MAT4	23	.092			.083		.020	.2	

Field	Description
MID	Material identification number. (Integer > 0)
Kii	Thermal conductivity matrix term. (Real ≥ 0.0)
CP	Thermal capacitance per unit volume. (Real ≥ 0.0 or blank)

Remarks

1. The material identification number must be unique with respect to all other MAT4 or MAT5 Bulk Data.
2. The thermal conductivity matrix has the following form:

$$[K] = \begin{bmatrix} K_{xx} & K_{xy} & K_{xz} \\ K_{yx} & K_{yy} & K_{yz} \\ K_{zx} & K_{zy} & K_{zz} \end{bmatrix}$$

3. Only the upper triangle of the conductivity matrix is specified
4. The **MATT5** is used in conjunction with a MAT5 to defined temperature-dependent properties.

MATT4 - *Thermal Material Temperature Dependence*

Description

Defines the thermal properties for temperature-independent, isotropic materials.

Format

1	2	3	4	5	6	7	8	9	10
MATT4	MID	T(K)							

Example

MATT4	23	73							

Field	Description
MID	Material identification number of a MAT4 for which the thermal conductivity is temperature-dependent. (Integer > 0)
T(K)	Identification number of a TABLEMi Bulk Data entity which defines the temperature-dependence of the Thermal conductivity or convective film coefficient. (Integer > 0)

Remarks

1. The thermal capacity can not be temperature-dependent.
2. **TABLEM1**, **TABLEM2**, **TABLEM4**, OR **MATT4** Bulk Data can be used. The value of K defined on the associated MAT4 is ALWAYS multiplied by the value obtained from the TABLEMi table.

MATT5 - *Thermal Material Temperature Dependence*

Description

Defines the thermal properties for temperature-independent, isotropic materials.

Format

1	2	3	4	5	6	7	8	9	10
MATT5	MID	T(KXX)	T(KXY)	T(KXZ)	T(KYY)	T(KYZ)	T(KZZ)		

Example

MATT5	23	73			74				

Field	Description
MID	Material identification number of a MAT5 for which the thermal conductivity is temperature-dependent. (Integer > 0)
T(Kii)	Identification number of a TABLEMi Bulk Data entity which defines the temperature-dependence of the matrix term. (Integer > 0)

Remarks

1. The thermal capacity can not be temperature-dependent.
2. TABLEM1, TABLEM2, TABLEM3, OR TABLEM4 Bulk Data can be used. The value of Kii defined on the associated MAT5 is ALWAYS multiplied by the value obtained from the TABLEMi table.
3. Blank or zero entries mean no table dependence on the associated property on the referenced MAT5 and the quantity is constant.
4. Material Properties on the MAT5 are initial values. If two or more quantities are to have a fixed relationship then two or more tables must be created that define the relationship.

Boundary Element - HBDY

A special element, the HBDY, defines an area to be associated with thermal boundary conditions. This element has five basic types called POINT, LINE, REV, AREA3, and AREA4. A sixth type, ELCYL is used only for radiation defined by QVECT. HBDY is considered to be an element since the temperature boundary condition adds terms to the Conduction Matrix.

CHBDY - *Heat Boundary Element*

Description

Defines a boundary element for heat transfer analysis which is used for heat flux, thermal vector flux and/or radiation.

Format

1	2	3	4	5	6	7	8	9	10
CHBDY	EID	PID	TYPE	G_1	G_2	G_3	G_4		
	GA_1	GA_2	GA_3	GA_4	V_1	V_2	V_3		

Example

CHBDY	721	100	LINE	101	98				
	102	102			1.0	0.	0.		

Field	Description
EID	Element identification number (Integer > 0)
PID	Property identification number (Integer > 0)
TYPE	Type of area associated with element. One of the following character values: POINT LINE REV AREA3 AREA4 ELCYL
G_1,G_2,G_3,G_4	Identification numbers of connected grid points (Integer > 0 or blank)
GA_1,GA_2,GA_3, GA_4	Grid or Scalar identification numbers of associated ambient points (Integer > 0 or blank)
V_1,V_2,V_3	Components of orientation vector defined in Basic Coordinates (real or blank)

Remarks

1. The continuation is NOT required
2. The six types have the following characteristics:

Table 7-3 CHBDY TYPE Characteristics

TYPE Name	Characteristics
POINT	Has one primary Grid Requires a PCHBDY The normal orientation vector is required if thermal flux vector is used

Table 7-3 CHBDY TYPE Characteristics

TYPE Name	Characteristics
LINE	Two primary Grid points Requires a PCHBDY The normal orientation vector is required if thermal flux vector is used
REV	Two primary Grid points which must lie in the x-z plane of the Basic coordinate system with x > 0. The defined area is that of a conical section with z as the axis of symmetry. A PHBDY is required for convection, radiation or thermal vector flux.
AREA3	Has three primary Grid points which define a triangular surface order to go around the boundary A PHBDY is required for convection, radiation or thermal vector flux.
AREA4	Has three primary Grid points which define a triangular surface order to go around the boundary A PHBDY is required for convection, radiation or thermal vector flux.
ELCYL	Has two primary Grid points A PHBDY is required if a thermal flux vector is defined The thermal flux vector must be nonzero.

3. The PHBDY property Bulk Data is used to define the associated area factors, the emmissivity, the absorbtivity and the principal radii of the elliptic cylinder. The material coefficients and the convection and thermal capacity are defined by a MAT4 of MAT5 which is referenced by the PHBDY.
4. The “Associated Points”, GA_1, GA_2, etc., may be either grid or scalar points and are used to define the ambient temperature when a convection field exists. These points correspond to the primary points, G_1, G_2, etc., and the number of the associated points depends on the TYPE option but they need not be unique. Their values may be set in static thermal analysis with SPC bulk Data, or they may be connected to other elements. If any of the fields is blank, the ambient temperature associated with the associated Grid point is zero. The associated degees of freedom should be unique among other CHBDY.
5. Heart flux may be applied to the element with QBDY1 or QBDY2 BUlk Data.
6. Thermal vector flux from a directional source may be applied to the HBDY element with QVECT Bulk Data. The orientation of the normal vector must be defined. The Grid point ordering establishes the normal direction as end “a” to end “b” for a LINE element and “right hand rule” for cross products elements. The Normal Direction for each element TYPE is shown by Figure XXX .

PHBDY - *Property of Heat Boundary Element*

Description

Defines properties of the HBDY element.

Format

1	2	3	4	5	6	7	8	9	10
PHBDY	EID	MID	A_F	E	ALPHA	R_1	R_2		

Example

PHBDY	100	103	300.	0.79					

Field	Description
PID	Property identification number (Integer > 0)
MID	Material identification number (Integer > 0)
A_F	Area Factor (Real ≥ 0.0 or blank). Used only for TYPEs LINE, POINT of ELCYL.
E	Emmissivity (1.0 ≥ E ≥ 0.0 of blank). Used for radiation calculations.
ALPHA	Absorbtivity (1.0 ≥ ALPHA ≥ 0.0 of blank). Used only for thermal flux calculations, default value is E.
R_1, R_2	“Radii” of elliptic cylinder. Used for TYPE ELCYL. (real)

Remarks

1. The referenced material ID must be that of a MAT4 which defines the convective film coefficient and thermal capacitance per unit area. If no material is referenced the element convection and capacitance are zero
2. The area factor, A_F, is used to determine the effective area. For TYPE POINT, AF is area. For TYPEs LINE and ELCYL, A_F is the effective width, where area = (A_F) (Length).The effective area is automatically calculated for other HBDY TYPEs.

Thermal Loads

Thermal loads may be boundary heat fluxes of volume heat addition. As in the case of structural analysis, the method of specifying loads is different for static an transient analyses.

The HBDY element is used for boundaries of conductive regions. Surface flux input can be specified on HBDY elements using **QBDY1** and **QBDY2** Bulk Data. These entities are for constant and spatially variable heat flux, respectively.

Flux can be specified without reference to HBDY with the **QHBDY** Bulk Data entity.

Vector flux, such as solar radiation, depends on the angle between the flux and the element normal, and is specified for the HBDY with the**QVECT** entity. This requires that the orientation of the HBDY be defined.

Volume heat addition is defined by **QVOL** Bulk Data.

QBDY1 - *Boundary Heat Flux Load*

Description

Defines a uniform heat flux into the HBDY element.

Format

1	2	3	4	5	6	7	8	9	10
QBDY1	SID	Q_0	EID	EID_i	THRU	EDI_m	EID_n		

Example

QBDY1	109	1.5e-5	721	723	THRU	790	799	801	
	850	901							

Field	Description
SID	Load set identification number (Integer > 0)
Q_0	Heat flux into the element (real)
EID	HBDY element identification number Integer > 0)

Remarks

1. QBDY1 must be selected by the LOAD Case Control command for static analysis. The power contributed by this entity is given by:

$$P_{in} = [(\texttt{effective area})Q_0]$$

where effective area is specified on the referenced PHBDY.

2. QBDY1 must be selected by the DLOAD Case Control command for transient analysis. The power contributed by this entity is given by:

$$P_{in} = [(\texttt{effective area})Q_0 + A]F(t - \tau)$$

where effective area is specified on the referenced PHBDY and *A* is specified by a DAREA Bulk Data entity and the time function, *F(t-τ)* is specified by TLOAD1 or TLOAD2 Bulk Data entities.

3. The flux, Q_0, is positive for input to the element.
4. Continuations are allowed.
5. EID may be specified as individual references or as a sequential list (“THRU” sequences) interchangeably. The only restriction is that the integer values must appear in fields 4 and 9 of the parent and in fields 2 and 9 of the continuation.

QBDY2 - *Boundary Heat Flux Load*

Description

Defines Grid point heat flux into the HBDY element.

Format

1	2	3	4	5	6	7	8	9	10
QBDY2	SID	EID	Q_{01}	Q_{02}	Q_{03}	Q_{04}			

Example

QBDY2	109	721	1.5-5	1.5-5	2.5-5	2.5-5			

Field	Description
SID	Load set identification number (Integer > 0)
EID	HBDY element identification number Integer > 0)
Q_{0i}	Heat flux at the ith grid point of the referenced HBDY element (Real or blank)

Remarks

1. QBDY2 must be selected by the LOAD Case Control command for static analysis. The power contributed by this entity is given by:

$$P_{in} = AREA_i \bullet Q_{0i}$$

where effective area is specified on the referenced PHBDY.

2. QBDY2 must be selected by the DLOAD Case Control command for transient analysis. All connected Grid points will have the same time function, but may have individual time delays, τ_i . The power contributed into each point, i, of the element by this entity is given by:

$$P_{in} = AREA_i \bullet Q_{0i} \bullet F(t - \tau_i)$$

where $AREA_i$ is specified by a DAREA Bulk Data entity and the time function, $F(t\text{-}\tau)$ is specified by TLOAD1 or TLOAD2 Bulk Data entities.

3. The flux, Q0, is positive for input to the element.

QHBDY - *Boundary Heat Flux Load*

Description

Defines a uniform heat flux into a set of Grid points.

Format

1	2	3	4	5	6	7	8	9	10
QHBDY	SID	TYPE	Q_0	A_F	G_1	G_2	G_3	G_4	

Example

QHBDY	109	LINE	1.5+3	.75	13	21			

Field	Description
SID	Load set identification number (Integer > 0)
TYPE	Type of area to be defined by Grid points; MUST be one of “POINT”, “LINE”, “REV”, “AREA3”, “AREA4”
Q_0	Heat flux into the element (Real)
A_F	Area factor depending on TYPE. (Real > 0.0 or blank)
G_1,G_2,G_3,G_4	Grid point identification numbers of connected points. (Integer > 0 or blank)

Remarks

1. The heat flux applied to the area is transformed to loads at the Grid points.
2. The flux applied to each point, i, is given by:

$$P_{in} = Q_0 \bullet AREA_i$$

where $AREA_i$ is the portion of the total area associated with point i.

3. For static analysis, the load SID of the QHBDY is selected by a LOAD Case Control command.
4. For transient analysis, the SID of the CHBDY must be selected by a DLOAD Case Control command. The load at each point is multiplied by a time function, $F(t-\tau_i)$, defined by TLOAD1 or TLOAD2, and where the time delay τ_i, is defined by DELAY Bulk Data.
5. The number of connected points for each TYPE is:

TYPE	Number of Points
POINT	1
LINE	2
REV	2
AREA3	3
AREA4	4

6. The area factor, AF, is used to determine the effective area for POINT and LINE TYPEs. It is equal to the area and the effective with, respectively. The field is ignored for other TYPEs.

7. The TYPE flag defines a surface in the same manner as the **CHBDY** element.

QVOL - *Volume Heat Addition*

Description

Defines a rate of internal heat generation in an element.

Format

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QV	EID_1	EID_2	EID_m	THRU	EID_n	EID_k	
	EID_r								

Example

QVOL	109	1.+2	310	303	317	THRU	345	416	
	527	623							

Field	Description
SID	Load set identification number (Integer > 0)
EID	HBDY element identification number (Integer > 0)
QV	Power input per unit volume produced by of the referenced HBDY element (Real or blank)

Remarks

1. QVECT must be selected by the LOAD Case Control command for static analysis. The effective power contributed by this entity into each grid point, i, is given by:

$$P_{in} = VOL_i \bullet QV$$

where VOL_i is the portion of the volume associated with the point, i, and QV is positive for heat generation.

2. QVECT must be selected by the DLOAD Case Control command for transient analysis that references a $TLOAD_i$ Bulk Data entity. All connected Grid points will have the same time function, but may have individual time delays, τ_i. The power contributed into each point, i, of the element by this entity is given by:

$$P_{in} = QV \bullet VOL_i \bullet F(t - \tau_i)$$

where VOL_i is the portion of the volume associated with point, i, and the time function, $F(t\text{-}\tau)$ is specified by TLOAD1 or TLOAD2 Bulk Data entities.

3. EID may be specified as individual element references or as sequential lists (“THRU” sequences) and the two forms can be used interchangeably. The only restriction is that integer values must appear in fields 4 and 9 of the parent and in fields 2 and 9 of each continuation.

QVECT - *Thermal Flux Vector Load*

Description

Defines a thermal flux vector from a distant source into a HBDY element

Format

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q_0	E_1	E_2	E_3	EID_1	EID_2	EID_3	
	EID_4	EID_5	EID_6						

Example

QVECT	109	1.-2	-1.0	0.0	0.0	310	345	416	
	527	623							

Field	Description
SID	Load set identification number (Integer > 0)
Q_0	Magnitude of thermal flux vector. (Real)
E_1,E_2,E_3	Components of the thermal flux vector in basic coordinates. (Real, integer > 0, or blank.)
EID_i	HBDY element identification number (Integer > 0)

Remarks

1. QVECT must be selected by the LOAD Case Control command for static analysis. The effective power contributed by this entity into each grid point, i, is given by:

$$P_{in} = -\alpha A(\vec{e} \bullet \vec{n}) \bullet Q_i$$

where:

a	Absorbtivity
A	Area of HBDY element
$\vec{e}$	Components of thermal flux vector
$\vec{n}$	Positive normal vector of element as described for **CHBDY**
$(\vec{e} \bullet \vec{n})$	Dot product, positive if flux is coming from behind the element.

2. QVECT must be selected by the DLOAD Case Control command for transient analysis that references a $TLOAD_i$ Bulk Data entity. All connected Grid points will have the same time function, but may have individual time delays, τ_i. The power contributed into the element by this entity is given by:

$$P_{in} = -\alpha A(\vec{e} \bullet \vec{n}) \bullet Q_0 \bullet F(t-\tau_i)$$

.where:

a	Absorbtivity
A	Area of HBDY element
$\vec{e}$	Components of thermal flux vector, each of which can be functions of time. If a field, E1, E2, or E3 is an integer then it is the identification number of a $TABLE_i$ defining the time variance; if the field is real it is a constant value; and, if it is blank, the component is zero.
$\vec{n}$	Positive normal vector of element as described for **CHBDY**
$(\vec{e} \bullet \vec{n})$	Dot product, positive if flux is coming from behind the element.
$F(t - \tau_i)$	Function the defines the time-variance of the load specified by TLOAD1 or TLOAD2 Bulk Data. The value of t is calculated for each loaded point.

3. If the referenced HBDY element is TYPE ELCYL, the power input is an exact integral over the area exposed to the thermal flux vector
4. If the referenced HBDY is of TYPE REV, the vector is assumed to be parallel to the Basic Z-axis.
5. A sequential list of elements can be defined by specifying the first element in field 8, “THRU”, in field 8, and the last element in field 9. If this option is used no subsequent EIDs are allowed.

Radiation Exchange

Radiation heat exchange may be included between HBDY elements. The list of elements must be specified on a RADLST Bulk Data entity. The emmissivities are specified by the associated PHBDY Bulk Data. The Stefan-Boltzman constant, SIGMA, and absolute temperature, TABS, are specified by PARAM Bulk Data. Radiation exchange coefficients, whose default is zero, are specified on RADMTX bulk Data.

The several types of power input to the HBDY can be output using the ELFORCE Case Control command.

RADLST - *List of Radiation HBDY Elements*

Description

Specifies a list of HBDY elements which will have radiation input energy.

Format

1	2	3	4	5	6	7	8	9	10
RADLST	EID_1	EID_2	EID_3	EID_4	EID_5	EID_6	EID_7	EID_8	
	EID_9	-etc.							

Example

RADLST	10	20	30	40	41	THRU	2050	2060	
	3000	4000							

Field	Description
EID	HBDY element identification number (Integer > 0)

Remarks

1. RADMTX is required to define the radiation matrix for the enumerated HBDY elements.
2. Only one RADLST is allowed in Bulk Data.
3. If the group of elements is sequential, any of the fields except 2 and 9 may contain “THRU”. Element IDs will be generated for all elements in the range of the “THRU”.
4. An element can be listed more than once. For example, if both sides of a panel are radiating, each side may participate in a different part of the radiation matrix.

RADMTX - *Radiation Exchange Matrix*

Description

Matrix of radiation exchange coefficients for nonlinear heat transfer analysis.

Format

1	2	3	4	5	6	7	8	9	10
RADMTX	INDEX	$F_{i,i}$	$F_{i+1,i}$	$F_{i+2,i}$	$F_{i+3,i}$	$F_{i+4,i}$	$F_{i+5,i}$	$F_{i+6,i}$	
	$F_{i+7,i}$	-etc.							

Example

RADMTX	0.0	9.3	0.0	17.2	16.1	0.1	0.0	6.2	
	6.2								

Field	Description
INDEX	Column number of the matrix (Integer > 0)
$F_{i+k,i}$	Row Values of INDEX column, starting at the diagonal and continuing down the column. A blank field ends the current column.

Remarks

1. The range of INDEX is 1 through NA, where NA is the number of radiating areas.
2. The radiation exchange matrix is symmetric so that only the diagonal and terms below the diagonal are entered. Column 1 is associated with the first HBDY entered on RADLST, Column 2 is associated with the second, etc. Null columns need not be entered.
3. The total irradiation for element i is given by:

$$P_i = \sum_{j=1}^{NA} F_{ij} q_j$$

where

Variable	Description
P_i	Total irradiation into the element
F_{ij}	Coefficients of the radiation exchange matrix, defined by RADMTX, whose unit is area.
q_j	Radiosity at j, whose unit is area

4. A columns can only be specified once.
5. An element appearing on RADLST for which there is no term in the RADMTX will cause the missing terms of the column to be filled with zeros. This implies that an infinite heat sink, i.e. radiation loss, is present.

Constraints an Partitioning

Thermal boundary conditions are applied as constraints which represent perfect conductors and provide other desired characteristics for the heat transfer model.

Single point constraints are used to specify the temperature at a point. SPC or SPC1 Bulk Data list the Grid or Scalar points for which temperatures are specified. Thermal constraints CANNOT be specified using the PS field of GRDSET or GRID Bulk Data.

In linear static analysis, the SPC and SPC1 are used to constrain nodes to a fixed temperature. In nonlinear analysis the SPC and SPC1 specify nodes to constrained. The actual value of the boundary temperature is specified on TEMP Bulk Data, which must be selected by a TEMP(MATERIAL) Case Control command. In transient analysis the SPC and SPC1 can be used to fix the temperature only when that temperature is zero.Otherwise a large conductive coupling to ground at absolute zero must be defines. The structural relationship, $F = Kx$, the thermal analogy is made where K is the conductive coupling. In the thermal case x is adjusted to the required temperature by defining K using a CELAS element, which is connected to ground, and a load F, which is applied to the node. The numerical value of K should be several orders of magnitude greater than those used for the other conductivities in the thermal model.

Multi point constraints are linear relationships between temperatures at several nodes. These relationships are defined using MPC Bulk Data.

Linear Steady State Analysis

Linear steady state analysis is specified in Executive Control by the statements:

```
APP HEAT
SOL 1
```

The subcase structure in Case Control allows many load and constraint conditions to be performed in a single NASTRAN-CORE execution.

Nonlinear Steady State Analysis

Nonlinear steady state analysis is specified in Executive Control by the statements:

```
APP HEAT
SOL 3
```

This solution supports temperature-dependent conductivities of the elements, nonlinear radiation exchange and a limited use of multipoint constraints. Since the solution is iterative only one set of loads and constraint conditions is allowed.

The following PARAMs are used for nonlinear thermal analysis:

PARAM Name	Description
MAXIT	Maximum number of iterations (Integer, Default = 4)
EPSHT	Solution convergence parameter (Real, Default = .001)
TABS	Absolute reference temperature (Real, Default = 0.0)
SIGMA	Stephan-Boltzman constant (Real, Default = 0.0)
IRES	Request residual vector output if positive (Integer, Default = -1)

The user must supply an estimate of the temperature distribution vector, $\{T^1\}$,using TEMP Bulk Data. This estimate is used to calculate the reference conductivity and radiation exchange matrix needed for the first iteration and is also used to define the boundary temperatures of all boundary points identified on SPC and SPC1 Bulk Data. The TEMP Bulk Data that specify $\{T^1\}$ must be selected by a TEMPERATURE(MATERIAL) Case Control directive.

The solution will be terminated for one of the following conditions:

1. Normal convergence: ε_T < EPSHT, where ε_T is the current error estimate
2. Number of iterations = MAXIT
3. Unstable: The stability parameter $|\lambda_1| < 1$.
4. Insufficient time to perform and additional iteration.

Error estimates for each iteration can be output by including the EXEC statement DIAG 18 in the Executive Control section.

Transient Analysis

Transient thermal analysis is specified in Executive Control by the statements:

APP HEAT
SOL 9

This algorithm can include conduction, film heat transfer, nonlinear radiation and NASTRAN-CORE NOLIN-type loads. All points to be loaded by NOLINs must be in the solution set. Extra points can be used to load the model with NOLINs. Loads may also be defined by TLOAD1 and TLOAD2 which load nodes associated with DAREA bulk data, The thermal static loads can also be modified by a time function for transient analysis. Time steps are specified by TSTEP Bulk Data.

The Case Control commands required for transient analysis are:

Case Control Directive	
TSTEP	Selects TSTEP Bulk Data
DLOAD	Selects loads
NONLINEAR	Select nonlinear loads defined by NOLIN Bulk Data
TEMP(MATERIAL)	Selected TEMP BUlk Data that specified the initial estimate of the thermal field

The PARAMs in transient analysis are:

:

PARAM Name	Description
TABS	Absolute reference temperature (Real, Default = 0.0)
SIGMA	Stephan-Boltzman constant (Real, Default = 0.0)
BETA	Integration factor for Houbolt method. (real, Default = .55)
RADLIN	Radiation is linearized if positive. (Integer, Default = -1)

Compatibility with Structural Analysis

Grid point temperatures for thermal stress analysis are specified on TEMP Bulk Data. These temperatures can be output on the Punch file using the Case Control Directive, THERMAL(PUNCH). Thus if the topology of the structural and thermal models are the same the output from the thermal analysis can be used directly as Grid point temperatures in the structural analysis.

8 PARAMETERS

In addition to the finite element model definition and selection capabilities found in the Executive Command, Case Control and Bulk Data sections of the input file, a number of other values, or parameters, may be set using the Bulk Data "PARAM" entry. Parameters are most often used to select global modeling conventions and options that frequently apply across boundary and load cases, and across element types. Further, their values are available for use directly in DMAP, in contrast to data which appear on commands and entries. Though their use is varied (as outlined below), what they do have in common is that specification at the individual grid, element, or subcase level would either be tedious or inappropriate, or both, hence their global nature.

This section lists all parameters used in NASTRAN-CORE, their types, defaults, and ranges of applicability. Many are solution sequence dependent. All, however, are selected via the Bulk Data "PARAM" entry, which may be repeated as many times as necessary, for as many parameters as need be specified

ASETOUT Optional in all rigid formats. A positive integer value of this parameter causes the ASET (or HASET) output data block to be generated by the GP4 module. A negative integer value or 0 suppresses the generation of this output data block. The default value is 0.

AUTOSPC Default =YES for all rigid formats. AUTOSPC specifies the action to take when obvious singularities exist in the stiffness matrix. AUTOSPC=YES means that obvious singularities will be constrained automatically. AUTOSPC=NO means that obvious singularities will not be constrained. If PARAM,AUTOSPC is set to YES, detected singularities with a ratio smaller than PARAM,EPZERO (default=1.E-6) will be automatically constrained with single point constraints or multiple point constraints. If PARAM,PRGPST is set to NO (default is YES), the table of the singularities is suppressed, except when AUTOSPC = NO. If AUTOSPC is set to YES and SPCGEN is set to a positive value, the program will create SPC entries in the punch file for all DOF which are constrained due to AUTOSPC detecting potential singularities. The resulting SPC entries may be used in future runs. Note: If you do intend to use these entries, be sure to verify that it is correct to constrain the DOF listed on them.

BAILOUT Default=0, refer to MAXRATIO

BETAD Optional in static analysis with differential stiffness. The integer value of this parameter is the number of iterations allowed for computing the load correction in the inner (load) loop before shifting to the outer (stiffness) loop, which adjusts the differential stiffness. The default value is 4 iterations.

COUPMASS Optional in all DISP rigid formats. A positive integer value of this parameter causes the generation of coupled mass matrices rather than lumped mass matrices for all bar elements, rod elements, and plate elements that include bending stiffness. This option applies to both structural and non structural mass for the following elements: CBAR, CONROD, CQUAD4, CROD, CTRIA3 and CTUBE. A negative value causes the generation of lumped mass matrices (translational components only) for all of the above elements. A value of -1 enables the ehnanced lumped mass formulation for solid elements, which should be more accurate than a conventional one (This is the default.), other elements currently use a conventional lumped mass for a value of -1. A value of -2 or less enables the conventional lumped mass. A zero value activates the following parameters.

CPBAR, CPROD, and CPTUBE Optional in all DISP rigid formats. These parameters are active only if COUPMASS = 0. A positive value causes the generation of coupled mass matrices for all elements of that particular type as shown by the following table:

Parameter	Element Types
CPBAR	BAR
CPROD	ROD, CONROD
CPTUBE	TUBE

A negative value (the default) for these parameters causes the generation of the lumped mass matrices (translational components only) for these element types.

CTYPE Required in rigid formats using the cyclic symmetry feature (DISP rigid formats 14 and 15). The value of this parameter (character string) defines the type of cyclic symmetry as follows:

(1) ROT - rotational symmetry.

(2) DRL - dihedral symmetry, using right and left halves.

(3) DSA - dihedral symmetry, using symmetric and anti symmetric components.

CURVPLOT Used only in transient response (SOL 9 and 12). A value of 0 or larger for this parameter is used to obtain SORT1 output in transient response. Integer, default = -1

CYCIO Optional in static analysis with cyclic symmetry (DISP rigid format 14). The integer value of this parameter specifies the form of the input and output data. A value of +1 is used to specify physical segment representation, and a value of -1 for cyclic transform representation. The default value is +1.

CYCSEQ Optional in rigid formats using the cyclic symmetry feature (DISP rigid formats 14 and 15. The integer value of this parameter specifies the procedure for sequencing the equations in the solution set. A value of +1 specifies that all cosine terms should be sequenced before all sine terms, and a value of -1 specifies alternating cosine and sine terms. The default value is -1.

DDRMM Integer - used to select data recovery method used in modal solutions. Default = 1

- -1 = Mode displacement
- 0 = Matrix method
- +1 = automatic select = default

EPSIO Optional in static analysis with differential stiffness. The real value of this parameter is used to test the convergence of the iterated differential stiffness. The default value is 1.0e-5.

EPZERO Test parameter for AUTOSPC.Real, default = 1.e-6

G Optional in the direct formulation of all DISPLACEMENT dynamics problems (DISP rigid formats 7, 8 and 9). The real value of this parameter is used as a uniform structural damping coefficient in the direct formulation of dynamics problems (see section 9.3.3 of the Theoretical Manual).

GRDEQ Optional in static and normal modes analyses (DISP rigid formats 1, 2, 3, 14, and 15). A positive integer value of this parameter selects the grid point about which equilibrium will be checked for the Case Control output request, MPCFORCE. If the integer value is zero, the basic origin is used. The default value is -1.

GRDPNT Optional in all DISP rigid formats. A positive integer value of this parameter causes the Grid Point Weight Generator to be executed. The value of the integer indicates the grid point to be used as a reference point. If the integer is zero (blank is not equivalent) or is not a defined grid point, the reference point is taken as the origin of the basic coordinate system. All fluid related masses are ignored. Additional details for the Grid Point Weight Generator are given in Section 5.5 of the Theoretical Manual. The following weight and balance information is automatically printed following the execution of the Grid Point Weight Generator.

(1) Reference point.

(2) Rigid body mass matrix [MO] relative to the reference point in the basic coordinate system.
(3) Transformation matrix [S] from basic coordinate system to principal mass axes.

(4) Principal masses (mass) and associated centers of gravity (X-C.G., Y-C.G., Z-C.G.). (5) Inertia matrix I(S) about the center of gravity relative to the principal mass axes.

(6) Inertia matrix I(Q) about the center of gravity relative to the principal inertia axes.
(7) Transformation matrix [Q] between S-axes and Q-axes.

INREL Integer parameter used in solutions 1 to determine if inertia relief will be used. Default = 1 - This parameter is used in addition to/instead of a SUPORT entry. If a SUPORT entry is present in SOL's 1 or 2, then conventional inertia relief is performed. If PARAM,INREL is set to -1 and a SUPORT is present, then conventional inertia relief is performed in SOL 1. If PARAM, INREL is set to -1 without a SUPORT entry or if it is set to -2, then automatic inertia relief is performed.

INRLM Integer parameter used in modal solutions to determine if inertia-based residual vectors will be generated. If INRLM is -1, inertia-based residual vectors will be calculated. Default = -1 (also see RESVEC and RESVINER)

INTERACT Optional in DISP static analysis (DISP rigid format 1). This parameter, like the SYS21 parameter, is of relevance only when your primary purpose is to make interactive restart runs. In such a case, the integer value of this parameter must be set to -1 (via a PARAM Bulk Data entry) in both the batch checkpoint run (that precedes the interactive restart run) as well as in the interactive restart run. If not so specified via a PARAM Bulk Data entry, the COMPOFF and COMPON instructions in the DMAP sequence that use this parameter assume a value of 0 for this parameter (see **Section 5.7**).

IPRTCI, IPRTCL, and IPRTCF Optional in static aerothermoelastic design/analysis of axial flow compressors (DISP rigid format 16). If IPRTi is a positive integer, then intermediate print will be generated in the ALG module based on the print option in the ALGDB data table. If IPRTi = 0 (the default), no intermediate print will be generated.

IRES Optional in all DISP statics problems. A positive integer value of this parameter causes the printing of the residual vectors following each execution of the SSG3 (or SSGHT) module.

ISTART Optional in direct and modal transient response (DISP rigid formats 9 and 12). A positive value of this parameter causes the second (or alternate) starting method to be used (see Section 11.4 of the Theoretical Manual). The alternate starting method is recommended when initial accelerations are significant and when the mass matrix is non-singular. The default value is -1 and causes the first starting method to be used.

K6ROT Real, default = 0.0 - optional scaling factor for in-plane rotational stiffness terms on QUAD4 and TRIA3 elements.

KINDEX Required in normal modes analysis with cyclic symmetry (DISP rigid format 15). The integer value of this parameter specifies a single value of the harmonic index. Higher KINDEX no. will result in getting higher mode.

KMAX Optional in static analysis with cyclic symmetry (DISP rigid format 14). The integer value of this parameter specifies the maximum value of the harmonic index. The default value is ALL which implies NSEGS/2 for NSEGS even and (NSEGS - 1)/2 for NSEGS odd.

LFREQ and HFREQ Default for LFREQ is -1. Default for HFREQ is 999999. These parameters are used in all modal formulations of DISP dynamics problems (DISP rigid formats 10, 11 and 12), unless LMODES is used. The real values of these parameters give the cyclic frequency range (LFREQ is the lower limit and HFREQ is the upper limit) of the modes to be used in the modal formulation. To use this option, parameter LMODES must be set to 0.

LMODES If the default (MLODES = 0) is taken all modes generated in the normal modes portion of the analysis will be used as modal coordinates. Available in all modal formulations of DISP and AERO dynamics problems (DISP rigid formats 10, 11 and 12).

MAXRATIO
BAILOUT Default = 1.+7. When the stiffness matrix is decomposed, the ratios of the original values on the diagonal to the values on the diagonal of the triangular factor matrix are an indication of potential mechanisms or instabilities. These ratios are calculated and compared to the value of MAXRATIO. If any of the ratios exceed this value, the run will be terminated unless PARAM,BAILOUT is set to -1 (which is not recommended). If any ratios exceed the value provided, a list of the offending DOF and their ratios will be provided by the program. When this list is printed, it provides the user with information about potential (and real) mechanisms or instabilities in the stiffness matrix. If there are mechanisms or instabilities, the dof listed are the last DOF in the matrix which might have prevented the problem. It is not recommended to simply constrain the listed DOF. Rather, if there is a problem in the model, run a static solution with a subcase for each listed DOF. In each subcase, constrain the associated DOF from the list to 1.0 units, while constraining all other DOF in the list to 0.0 displacement. The resulting deformed shapes will allow visualization of the potential mechanisms or instabilities. If PARAM,BAILOUT is set to -1 (default is 0), the program will ignore potential stabilities in the matrix and attempt to provide a solution. Any solutions obtained with this PARAM,BAILOUT,-1 should be checked carefully to ascertain whether the solutions are valid.

PARAM,MECHOUT,YES will perform data recovery on the potential singularities, which can assist in determining whether they should exist or not.

MECHOUT BCD - default = YES. If matrix decomposition fails with terms which exceed MAXRATIO, this parameter controls the printout of the potential mechanisms/rigid-body modes. The default value will use the output requests in the case control to print displacement, SPC forces, element stresses, and element forces for each of the potential shapes found. A value of NO for this will disable this feature.

MODACC Optional in the modal formulation of frequency response (DISP rigid format 11) and transient response (DISP rigid format 12) problems. A positive integer value of this parameter causes the Dynamic Data Recovery module to use the mode acceleration method. Not recommended for use in hydro elastic problems. DMAP module GKAD sets the V1 value of PARAM MODACC to +1 for rigid format 12, and to -1 for rigid format 11.

NINPTS Optional in DISP static analysis (DISP rigid format 1). A positive integer value of this parameter specifies the number of closest independent points to be used in the interpolation for computing stresses or strains/curvatures at grid points (only for TRIA1, TRIA2, QUAD1 and QUAD2 elements). A negative integer value or 0 specifies that all independent points are to be used in the interpolation. The default value is 0.

NLOAD Optional in static analysis with cyclic symmetry (DISP rigid format 14). The integer value of this parameter is the number of static loading conditions. The default value is 1.

NONCUP Optional in transient and frequency response analysis. This integer parameter is set by GKAM. A negative value indicates that the uncoupled solution is to be used. Default = +1

NSEGS required in rigid formats using the cyclic symmetry feature (DISP rigid formats 14 and 15). The integer value of this parameter is the number of identical segments in the structural model.

NT Optional in static analysis with differential stiffness. The integer value of this parameter limits the cumulative number of iterations in both loops. The default value is 10 iterations.

OFFSET	a user warning message will be printed if the offset length of a BAR element exceeds 15 percent of the bar length. This default value of 15 percent can be changed using a PARAM, OFFSET entry.
OLDGP4	Integer, default = -1 - used in GP4 module to determine processing method - the default value (-1) uses the V3 GP4 logic (multi-point constraint equations in RG in the order they are processed), a value of +1 uses the old (pre-V3) logic (multi-point constraint equations arranged in order of ascending M-set dof) NOTE: the older method does not allow redundant equations.
OLDHEXA	Integer, default = 0 - a non-zero value for this parameter will force the program to use the old (pre-V2 HEXA formulation)
OLDMCE1	Integer, default = -1 - selects the processing approach used in MCE1 module. The default (-1) uses the V3 logic (automatic M-set selection and proper handling of redundant MPC equations). A value of +1 will select the old (pre-V3) logic.
OLDPENTA	Integer, default = 0 - a non-zero value for this parameter will force the program to use the old (pre-V2 PENTA formulation)
OLDQUAD4	Integer, default = 0 - a non-zero value for this parameter will force the program to use the old (pre-V2 QUAD4 formulation)
OLDTETRA	Integer, default = 0 - a non-zero for this parameter will force the program to use the old (pre-V2 TETRA formulation)
OPT	Optional in static and normal modes analyses (DISP rigid formats 1, 2, 3, 14, and 15). A positive integer value of this parameter causes both equilibrium and multipoint constraint forces to be calculated for the Case Control output request, MPCFORCE. A negative integer value of this parameter causes only the equilibrium force balance to be calculated for the output request. The default value is 0 which causes only the multipoint constraint forces to be calculated for the output request.
OUNIT2	Integer, default=12, refer to PARAM,POST
P1, P2, and P3	See the DMAP User's Manual for a description of these parameters required by the INPUTT2 module. The default values for P1, P2 and P3 are 0, 11 and "XXXXXXXX", respectively.
POST (OUNIT2)	Integer, default=0, If PARAM,POST is set to a negative number, the model information and analysis results will be written to a file (OUNIT2) using the OUTPUT2 module. Many post-processors can read the information from this file to allow users to view the model and results. PARAM,OUNIT2 sets the unit number of FORTRAN file used by OUTPUT2. The default for PARAM,OUNIT2 is 12. When PARAM,OUNIT2 is not set to other value, NASTRAN-CORE will automatically create the "op2" file using FORTRAN unit 12 for OUTPUT2 processing. Any other unit numbers will need to have a file name associated with them when submitting the program.
PRGPST	Controls print of singularity table generated during singularity processing to <filename>.gpst. Table will be printed if set to YES. Default = NO. See AUTOSPC.
PTHRESH	Threshold on applied load printout (applicable only for substructuring). Real, default = 0.0
QTHRESH	Threshold on SPC force printout (applicable only for substructuring). Real, default = 0.0
RESVEC	Character parameter used in modal solutions - if this parameter has a value of "YES", then loading-based residual vectors will be calculated. Default = YES (Also see RESVINER adn INRLM)
RESVINER	Character parameter used in modal solutions to determine if inertia-based residual vectors will be calculated. If this parameter has a value of "YES", then inertia-based residual vectors will be calculated. Default = YES (also see RESVEC and INRLM)
SDRDENS	Integer, default = 50 - used in SPARSEDR module - if the density (&) of PVGRID is less than SDRDENS, then the sparse method of data recovery will be selected. Otherwise, the full method will be selected. See SDROVR for overriding these actions.

SDROVR Character - default = AUTO - used to override the automatic selection of data recovery approach in the SPARSEDR module. If SDROVR = SPARSE, the module will select the sparse data recovery approach, if SDROVR = FULL, it will select the full (standard) data recovery approach, and if it is AUTO, the module will automatically select the approach based on the output requests and the value of SDRDENS.

SINGOPT Controls how singularities are removed during constraint processing. The two acceptable values are SPC (default) or MPC. If SPC then displacement dof whose displacement coordinate is closest to the to principal direction of the singularity will be constrained. If MPC a set of constraint equations will be generated. See AUTOSPC.

SOLTYP Integer - default = 1 in SOL 1 and 2, 0 in all other SOL's. If this parameter is set to 1, then all GAP dof (defined by the GAP entry), which are not placed in the S-set by the user are automatically treated as if they were on ASET entries (forcing a static of guyan reduction). A value other than 1 for this parameter allows GAP dof which are not placed in the A- or S-set by the user to be placed in other sets by the MCE1 module when it performs the automatic M-set selection.

SPCGEN A positive value results in the writing of SPC bulk data entries for SPC's generated during automatic constraint processing. Default = -1. See AUTOSPC

STRAIN Optional in DISP static analysis (DISP rigid format 1). This parameter controls the transformation of element strains/curvatures to the material coordinate system (only for TRIA3 and QUAD4 elements). If it is a positive integer, the strains/curvatures for these elements are transformed to the material coordinate system. The default value is -1.

STRESS Optional in DISP static analysis (DISP rigid format 1). This parameter controls the transformation of element stress output to the material coordinate system (only for TRIA3 and QUAD4 elements). If it is a positive integer, the stress otuput for these elements are transformed to the material coordinate system. The default value is -1.

SURFACE Optional in all DISP rigid formats. The computations of the external surface areas for the two-dimensional and three-dimensional elements are activated by this parameter when they are generated in the EMG module. The results are multiplied by the real value of this parameter. See the VOLUME parameter below for the case where the surface areas are to be saved on an output file. The surface areas of the three-dimensional elements are defined as follows.

Surface Area No.	Corner Grid Points Used
Brick (8 or more grid points):	
1	1, 2, 3, 4
2	1, 2, 6, 5
3	2, 3, 7, 6
4	3, 4, 8, 7
5	4, 1, 5, 8
6	5, 6, 7, 8
Wedge (6 grid points):	
1	1, 2, 3
2	1, 2, 5, 4
3	2, 3, 6, 5
4	3, 1, 4, 6
5	4, 5, 6
Tetrahedron (4 grid points):	
1	1, 2, 3
2	1, 2, 4
3	2, 3, 4
4	3, 1, 4

SYS21 Optional in DISP static analysis (DISP rigid format 1). This parameter, like the INTERACT parameter, is of relevance only when your primary purpose is to make interactive restart runs. In such a case, the integer value of this parameter must be set to -1 in the interactive restart run (that follows a batch checkpoint run). If not, the COMPOFF and COMPON instructions in the DMAP sequence that use this parameter assume a value of 0 for this parameter.

TABS Optional in nonlinear static (HEAT rigid format 3) and transient (HEAT rigid format 9) heat transfer analyses. The real value of this parameter is the absolute reference temperature. The default value is 0.0.

TINY In ESE, EKE, and EDE output, any element whose percentage of the total energy is less than TINY will not be included. Note: When used, this may cause unexpected gaps in contour plots using post-processors.

USETPRT Integer, default = -1 - Optional in all solutions - it controls the printout of the degree-of-freedom set definition tables

A value of 0 for USETPRT will result in a tabular printout using column sort of the SB, SG, L, A, F, N, G, R, O, and M sets.

A value of 1 will result in a row sort printout of the M, S, O, A, R, SG, and SB. Optionally, you may choose up to 4 sets to be printed using this format by using parameters, USETSTRi (i = 1-4)

A value of 2 will result in both output.

UTHRESH Threshold on displacement, velocity, and acceleration printout (applicable only for substructuring).

Real, default = 0.0

VOLUME Optional in all DISP rigid formats. The volume computations for the two-dimensional and three-dimensional elements are activated by this parameter when they are generated in the EMG module. The results are multiplied by the real value of this parameter. If the 7th output data block of the EMG module is specified (via DMAP ALTER), the element IDs, volumes, surface areas (see the SURFACE parameter above), SIL, and grid point coordinates are saved in the data block, a GINO-written file. If the 7th output data block is one of the INPi (i=1,2,3,...,9,T) files, the same element data is saved on a FORTRAN (binary)-written file. The following table summarizes the data being saved.

Record	Words	Contents
0	1,2	Header record, begins with GINO name (character)
	3-34	Title, character
	35-66	Sub-title, character
	67-98	Label, character
	99-101	Date, character
1	1,2	Element name of the first element, character
	3	Element ID, integer
	4	Volume (multiplied by scale factor n), or zero, real
	5	(No. of surfaces)*100 + (No. of grid points), integer
	6	Surface area of first surface, real :
	5+N	Surface area of N-th surface, real
	5+N+1	SIL of the first grid point, integer
	5+N+2,3,4	x,y,z coordinates of the first grid point, real:
		Repeat last 4 words for other grid points
2		A record similar to record 1 for the second element : :
LAST		Last record (for the last element).

The trailer of the output data block has the following information: Word 1 = LAST (No. of records written, header excluded), Words 2 through 6 contain no useful information.

W3 and W4 Optional in the direct formulation of DISP transient response problems (DISP rigid format 9). The real values (radians/unit time) of these parameters are used as pivotal frequencies for uniform structural damping and element structural damping, respectively (see Section 9.3.3 of the Theoretical Manual). Parameter W3 is required if uniform structural damping is desired. Parameter W4 is required if structural damping is desired for any of the structural elements. Parameter W3 should not be used for hydro elastic problems.

WTMASS Optional in all DISP rigid formats. The terms of the structural mass matrix are multiplied by the real value of this parameter when they are generated in the EMA module.